Repair Assessment Procedure And Integrated Design

RAPID

FOR

COMMUTERS

Version 1.2

User's Manual

March 2001

Rapid Disclaimer

The development of the **RAPID** software program was funded under contract by the Federal Aviation Administration (**FAA**) and the **USAF** and is provided to the user as is. Neither the **FAA** nor the U.S. Government warrants the merchantability of this software or its fitness for a particular use. Use of **RAPID** or its output does not replace or change any part of the repair or alteration approval process. Users of **RAPID** must obtain approval of any repair or alteration designed or analyzed with **RAPID** with the same procedures as with any other analysis tool. A repair design from **RAPID** should never supercede that supplied by the original equipment manufacturer (OEM). **RAPID** should be considered an alternative design tool whenever a repair procedure is not provided or is not available from the OEM. In no instance will the U.S. Government be liable for any damages sustained by the user, or any third party, caused by or attributed to the use of the **RAPID** software.

Table of Contents

1. Introduction

This document provides installation and usage instructions for the Repair Assessment Procedure and Integrated Design (RAPID) program. **This version has a completely new look and feel**. All users should read this manual.

RAPID is a tool for the design and analysis of aircraft structural changes. This version assists the user in designing an antenna installation as well as the mechanically fastened fuselage skin repairs with up to three doublers. RAPID performs both static and damage tolerance analyses of the repair. The static analysis determines if the mounting plates or doublers and fasteners are statically adequate, while the damage tolerance analysis yields crack growth life and residual strength.

2. Requirements

The following are the minimum hardware and software requirements for installing and running the RAPID software:

- Pentium class CPU
- Windows 95, Windows 98, Windows NT 4.0
- Hard disks with 100 megabytes free disk space. The load spectra, if generated, require an additional 11 megabytes each.
- 32 megabytes random access memory (RAM) (64 megabytes or more is preferred).
- 3.5 inch 1.44MB floppy disk drive.
- Super VGA, 800 by 600 dpi resolution (1024 by 768 or higher is recommended).
- Mouse or track-ball recommended.

3. RAPID Program Notes and Limitations

This section lists graphical user interface notes and program limitations for the current version of RAPID. Updated versions of this information will be available in RAPID from the **Readme Information** menu item in the **Help** pull-down menu.

4. Document Conventions

This manual uses the following special fonts:

ALL CAPS	This type represents file categories or names and disk directories
Bold	This type represents application commands, menu items, edit fields, etc.
Italic	This type represents string(s) that are found in a file
`Monospace`	This type represents text that user types or text as it appears on the screen.

5. RAPID Support Services

5.1 Telephone Support

For technical assistance, problem reporting, or general questions regarding RAPID, please contact:

RAPID Project Manager
FAA William J. Hughes Technical Center
Airframe Structures Section, AAR-431
Atlantic City International Airport, NJ 08405
http://aar400.tc.faa.gov/rapid

Any problems or difficulties encountered while running RAPID should be reported. If you discover any problems with RAPID, fill out the RAPID Problem Description Form supplied in the subsequent section documenting the events that led up to the problem in RAPID and the manifestations of the problem. This form should be submitted as soon as the problem is discovered. The same information may be supplied using e-mail if the user so desires.

5.2 RAPID Problem Description Form

Name: _____ Phone: _____

Company: _____ FAX: _____

Address: _____ e-mail: _____

City, State & Zip Code: _____

Problem Description:

_____
_____
_____
_____
_____
_____
_____
_____
_____
_____

FAA Use Only

Date Received: _____ Date Validated: _____ Date of Response: _____
Problem Source (Circle One)

Application
Documentation
Hardware
System Software
Other: _____

Module Affected: _____ Date Fixed/Incorporated: _____

Final Disposition:

_____
_____
_____
_____
_____

6. Installation

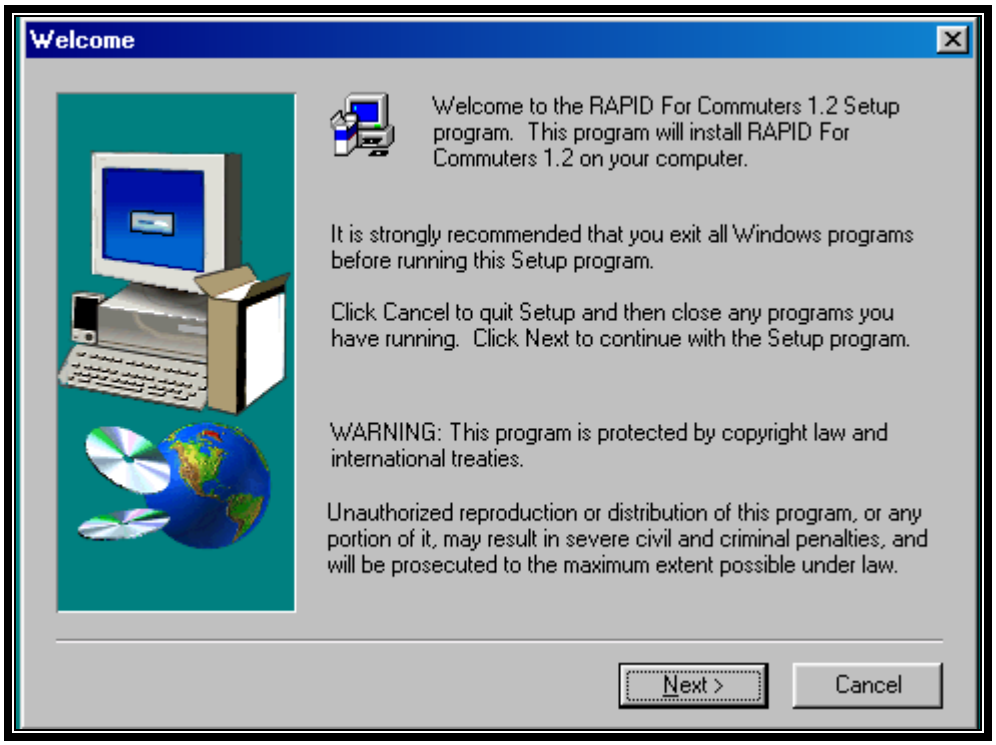
6.1 Run Setup

Exit all other Windows programs, insert Disk 1, and then run the `setup` program. Setup can be run by double-clicking the icon or by entering the following command at a DOS prompt or in a "Run…" dialog.

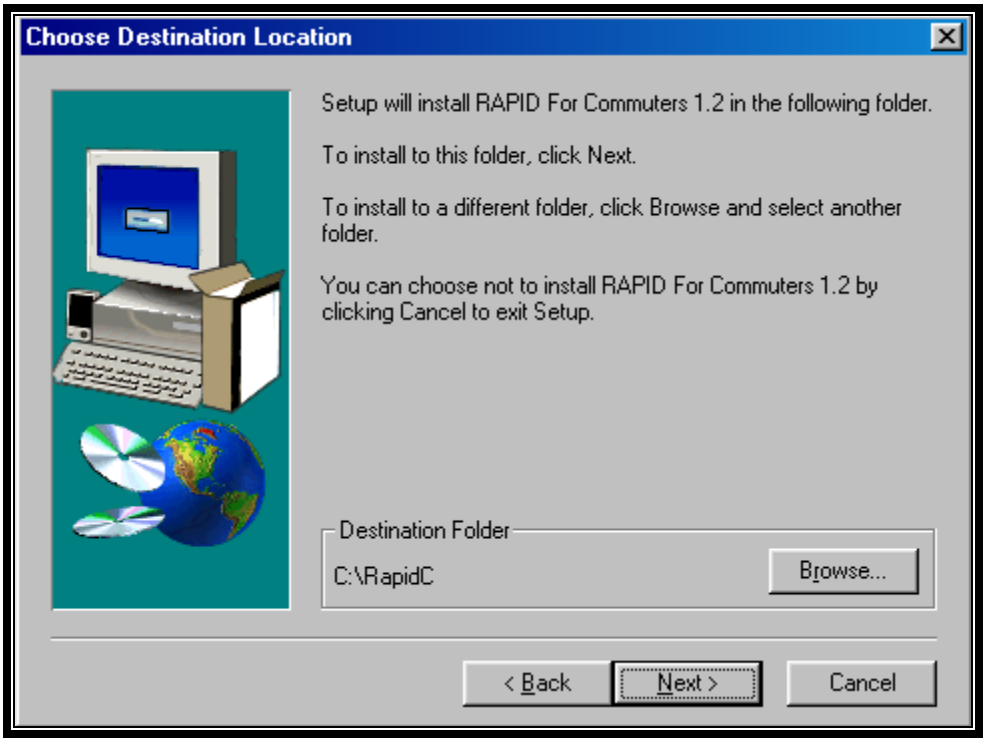
```
a:\setup
```

If a previous version of RAPID is already installed, then user created models and data files should first be backed up, and then the previous version of RAPID should be uninstalled or removed. Older versions of RAPID which do not include an uninstall utility can be removed by simply deleting the RAPID directory and its contents. Please note that repair configuration data files generated by older versions of RAPID may not be compatible with the current version.

After a few moments, the following dialog will appear. Select the **Next** button to continue.



6.2 Choose Destination Location

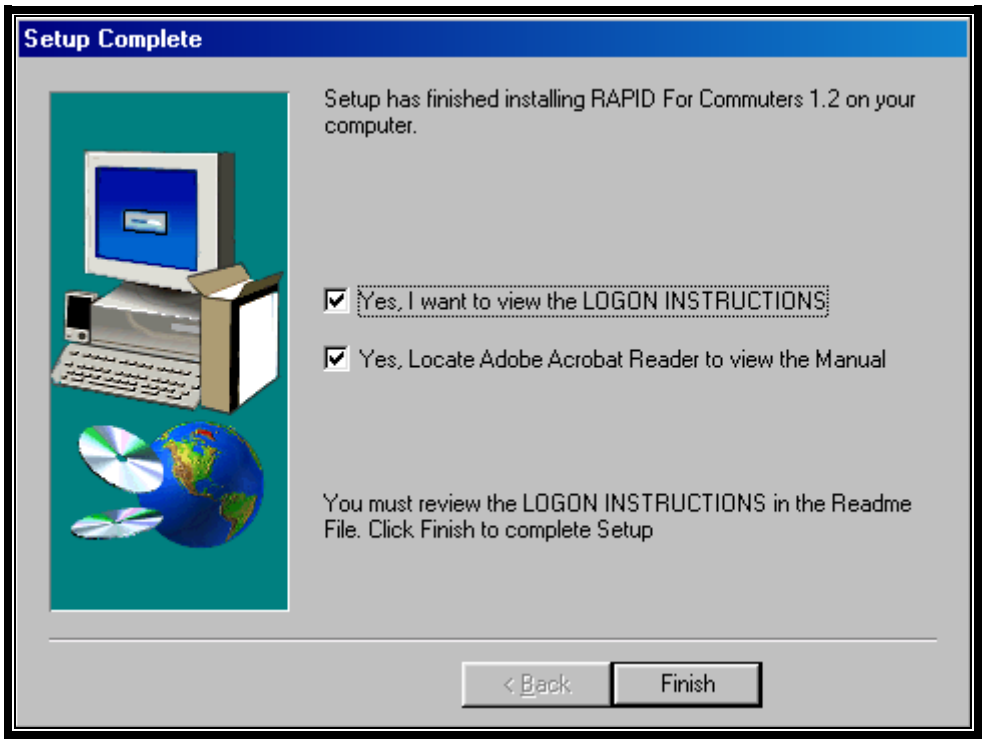


To install RAPID in the default `C:\RapidC\` directory, select the **Next** button. To install elsewhere, select the **Browse** button and enter an alternate directory location.

When installing from disk, a dialog will appear which will prompt the insertion of the next disk. Simply remove the current disk, insert the requested disk, and press the **Enter** key on the keyboard.

6.3 Complete the Installation

The following dialog will appear to verify that the installation of RAPID is complete. Click the **OK** button to exit the installation procedure. Remove the remaining disk from the floppy drive and store the disks in a safe location.



The RAPID program may then be run by double-clicking on the installed **RAPID FOR COMMUTERS 1.2** program icon. In Windows 95, 98 and Windows NT 4.0, RAPID may also be run by selecting it from within the **Programs** section of the Windows **Start** button.

6.4 Change to 1024 by 768 mode (recommended)

Although it is possible to use RAPID with any Windows-compatible display, it is recommended that a minimum window resolution of 800 by 600 dpi be used so that the windows will fit the display without the use of scrolling. However, 1024 by 768 or higher provides space to easier work on multiples configurations. For Windows 95, 98 and Windows NT 4.0, please refer to the hardware and operating system documentation included with the computer. Typically, the video mode can be changed through the **Settings** tab on the **Display Properties**, accessible from both the **Control Panel**, or by clicking the right mouse button on the Windows desktop and then selecting **Properties**.

6.5 Program Setup

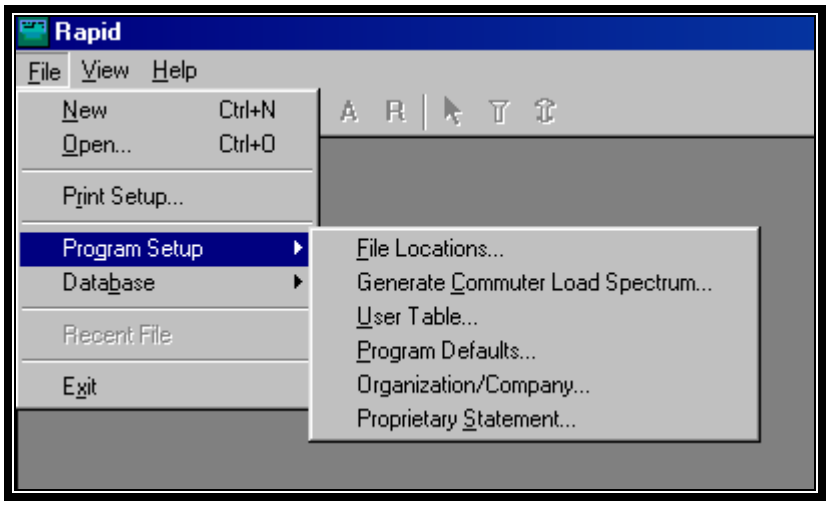


Figure 6-1

6.5.1 File Locations

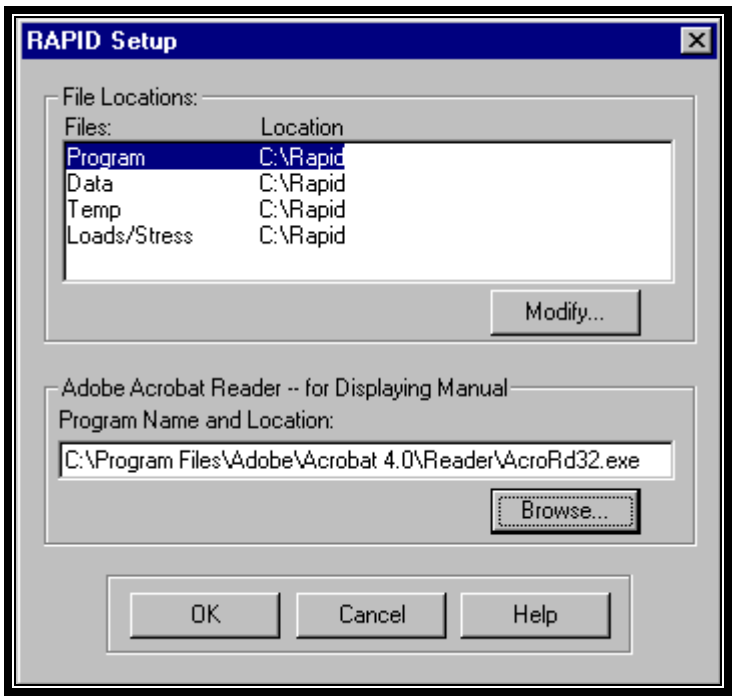


Figure 6-2

The data in this dialog can only be changed by the Administrator.

This dialog is accessed from the menu bar with File | Program Setup | File Locations.

The file locations are originally determined at the time of the installation. These locations can be modified by selecting the File to be changed. Mouse click on the “Modify…” button to choose a new Location (subdirectory).

At this time, the Adobe Acrobat Reader must be entered by the user. The program will attempt to locate the Adobe Acrobat subdirectory. But, because there are so many naming conventions, the user must use the “Browse” button to select the correct executable.

It is used to display and print the this Manual.

6.5.2 Generate Commuter Load Spectrum

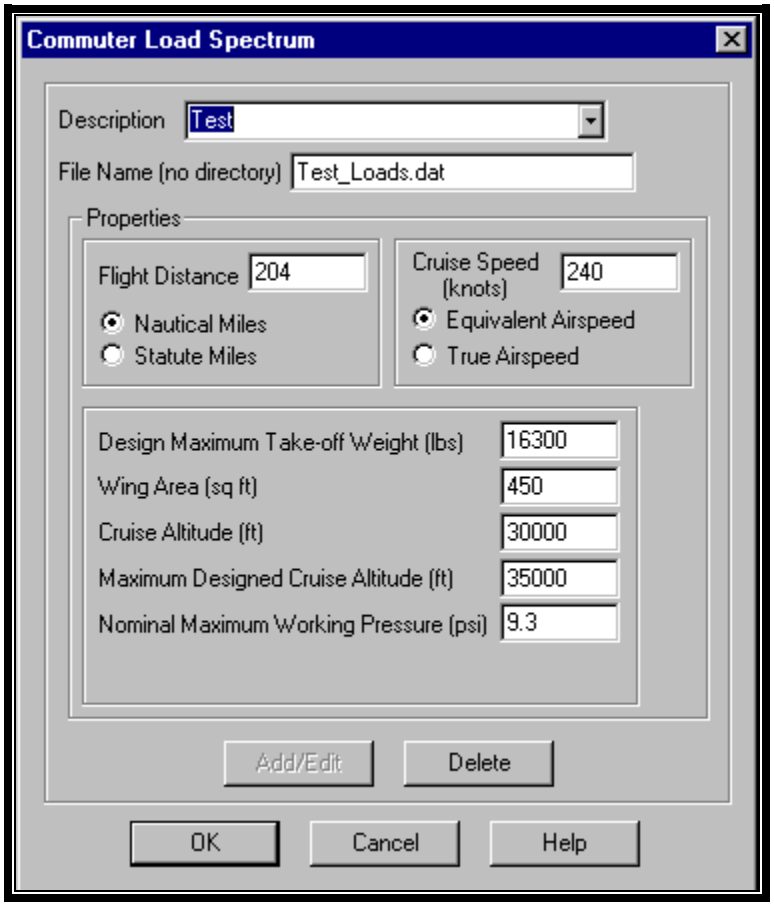


Figure 6-3

The data in this dialog can only be changed by the Administrator.

This dialog is accessed from the menu bar with File | Program Setup | Generate Commuter Load Spectrum.

It must be used if the user is going to utilize the capabilities in RAPID with respect to commuter load spectra. This will generate the load spectrum that is used later to generate the stress spectra and if required the equivalent stress. Any load spectrum created will be stored in the directory where RAPID resides.

To **Add** a new spectrum:

1. Start typing in the **Description** edit field.
2. Give the new spectrum a **File Name** (no directory --- the program File Locations places it in the correct subdirectory).
3. **Add** the properties (using the limits listed below);

The input parameters to generate a commuter load spectrum are as follows:

1. **Flight Distance** (Miles) ---- Minimum: 1.0 nautical miles, Maximum: 2,500 nautical mile
    a. **Nautical Miles**
    b. **Statute Miles** ---- 1.0 statute mile = 0.8684 nautical mile
2. **Cruise Speed** (Knots) ---- Minimum: 50 Knots (true airspeed), Maximum: 500 Knots (true airspeed)
    a. **Equivalent Airspeed** ---- program will calculate the true airspeed with given cruise altitude.

b. **True Airspeed**

3. **Design Maximum Take-off Weight (lbs.)** ---- Minimum: 4,000 lbs., Maximum: 19,000 lbs.
4. **Wing Area (sq.ft.)** ---- Minimum: 150 sq.ft., Maximum: 450 sq.ft.
5. **Cruise Altitude (ft.)** ---- Minimum: 100 ft., Maximum: 45,000 ft.
6. **Maximum Cruise Altitude (ft.)** ---- Maximum 45,000 ft.
7. **Nominal Maximum Working Pressure (psi)** ---- Minimum: 0 psi, Maximum: 10 psi

To **Edit** a Commuter Load Spectrum; select a spectrum from the **Description** combo box. Make the appropriate changes in the Properties edit fields. Mouse click the **Edit** button.
NOTE: If you want to change the File Name; you first must delete the spectrum; then create a new spectrum.

To **Delete** a Commuter Load Spectrum; select a spectrum from the **Description** combo box. Mouse click the **Delete** button.

6.5.3 User Table

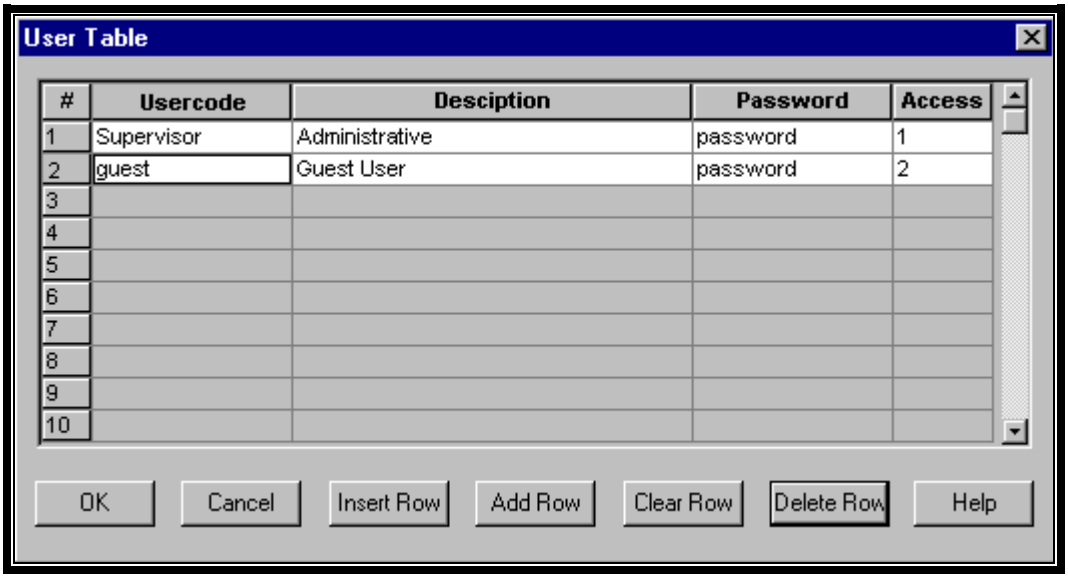


Figure 6-4

This table is for use by the Administrator Only.

NOTE: The Supervisor **Password** needs to be changed after receipt of the program

Usercode --- up to 12 alpha numeric characters
Description --- up to 25 alpha numeric characters
Password --- up to 10 alpha numeric characters
Access --- 1 is Administrator level --- can make changes in the Program Setup. 2 is User level --- has read only to the Program Setup

Insert Row --- Select the row that you want to Insert Row ahead.
Add Row --- Select the row that you want to Add Row after.
Clear Row --- Clears a row of all entries.
Delete Row --- Delete a row.

The changes are not saved until **OK** is clicked. **Cancel** will return the User Table to its last saved state.

6.5.4 Program Defaults

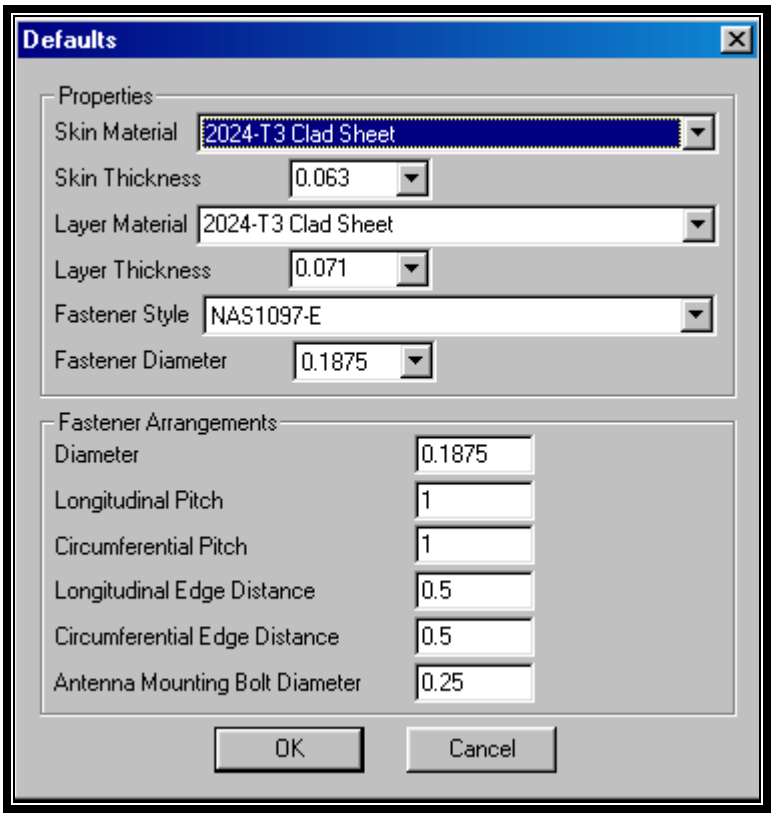


Figure 6-5

The data in this dialog can only be changed by the Administrator.

This dialog is accessed from the menu bar with File | Program Setup | Program Defaults.

This dialog allows the administrator to set default values for various design details that are utilized in the repair.

6.5.5 Organization / Company Info

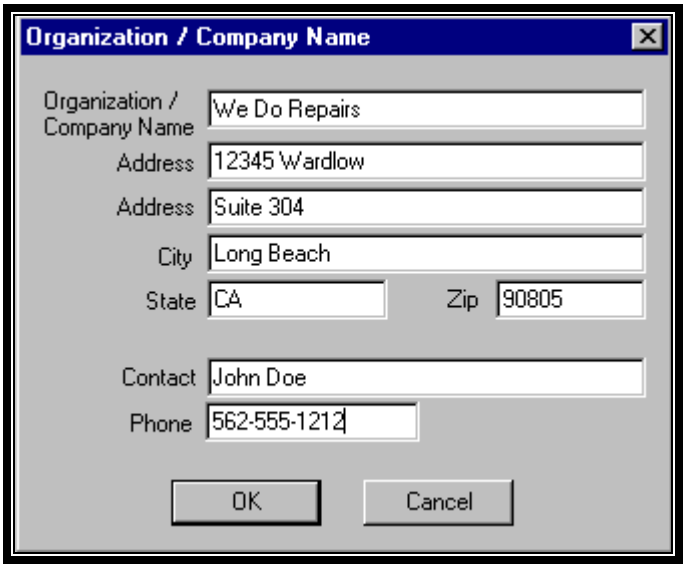


Figure 6-6

This dialog is accessed from the menu bar with **File | Program Setup | Organization/Company Info**.

This dialog allows the administrator to record the **Organization / Company Info** , some of which will be used on the **Report**.

6.5.6 Proprietary Statement

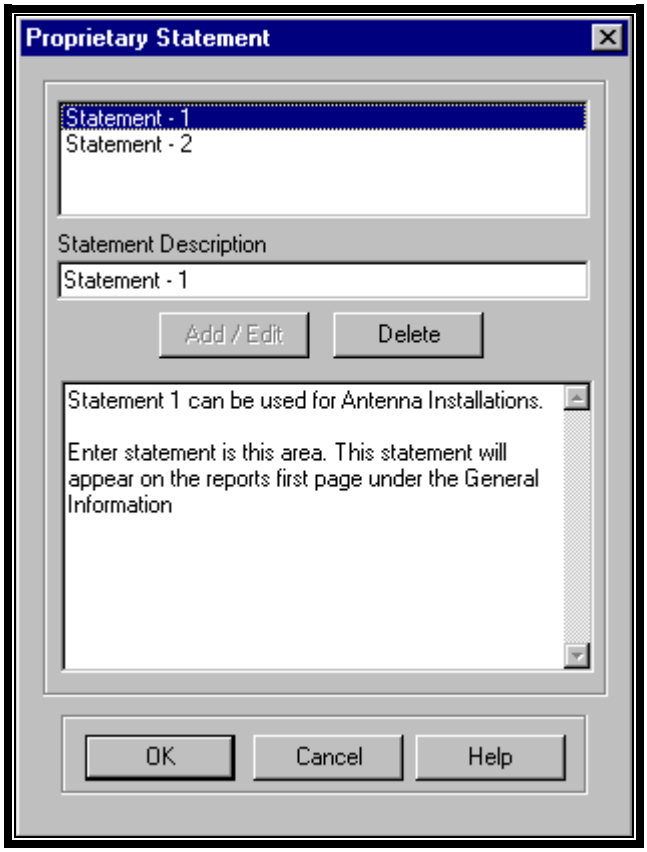


Figure 6-7

The data in this dialog can only be changed by the Administrator.

The Proprietary Statement is used as a field in the Report. It is not a required field.

The first time this page is used, sample statements will be present.

The top box is a list box containing a list of existing statement descriptions. It is a non-editable field. Picking a statement in this box will cause the statement information to be displayed in the other 2 edit fields.

The middle edit field is a **Description** of the statement to be used in the list box above.

The bottom edit box is a multi-line edit field that can be treated as a mini editor for entering the text of the **Statement.**

"**Add**" --- To add a statement, start typing in **Description** or **Statement** fields. When finished, "**Add**" the statement by clicking on the "**Add/Edit**" button. The description will be added the list box.

"**Edit**" --- Only the Statement can be edited, the Description cannot be edited. Make the appropriate changes in the statement and mouse click on the "**Add/Edit**" button

.

To edit the **Description**, start typing the **Description** field. When finished, mouse click on the "**Add/Edit**" button. This new **Description** will be added to the box. Then "**Delete**" the previous Description.

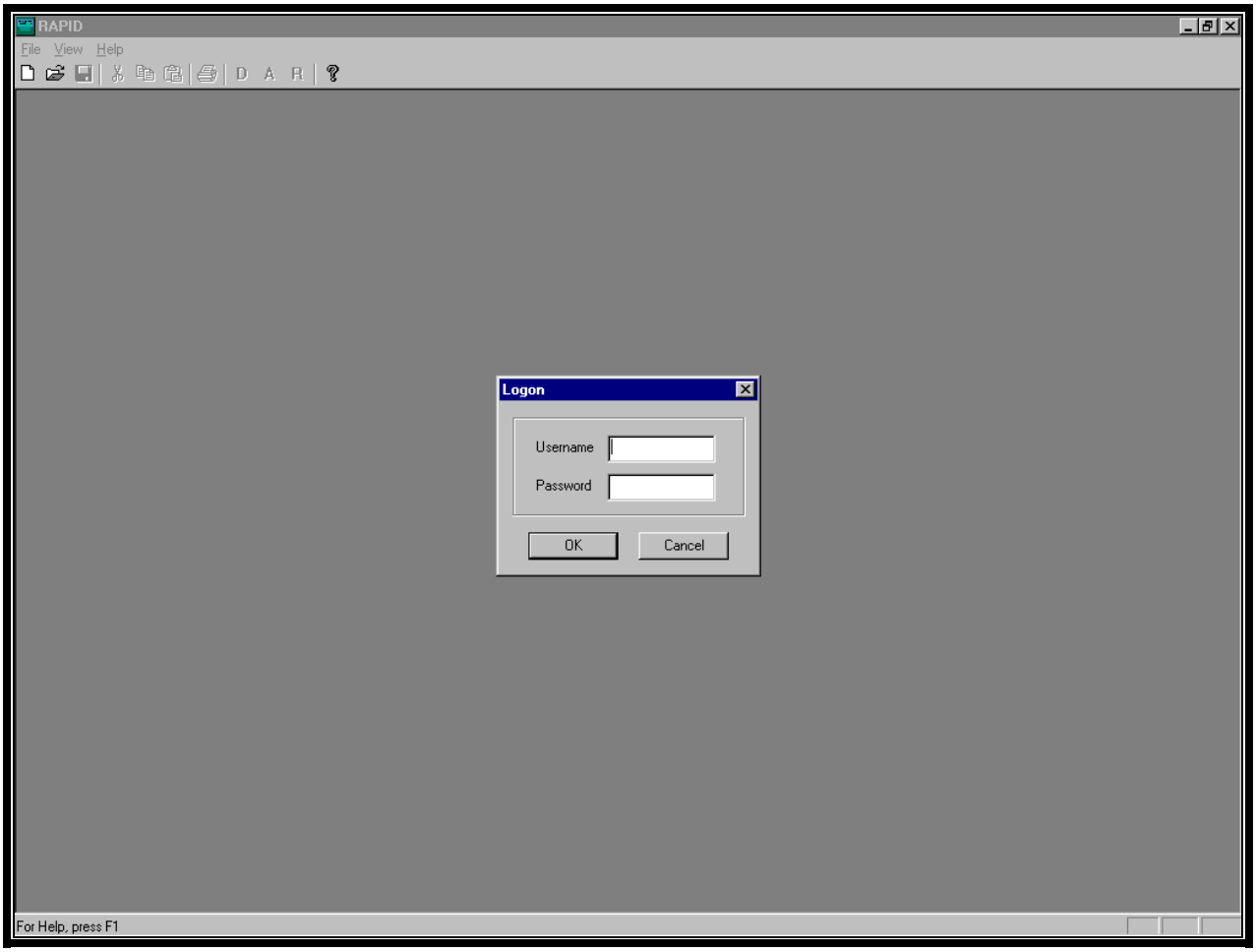
"**Delete**" --- Mouse click on the **Description** in the top list box; then mouse click the "**Delete**" button.

7. Running RAPID

RAPID is run by double-clicking on the installed **RAPID For Commuters 1.2** program icon. RAPID may also be run by selecting it from within the **Programs** section of the Windows **Start** button. When opened, RAPID fills the screen with the RAPID Multi-Document Interface (MDI). Commonly used tools and options are available both from the icon button bar and from the pull-down menus. Other options are available only from the pull-down menus. As with most Windows applications, the pull-down menus can be accessed both by mouse selection and by holding down the "Alt" key then pressing the key-character of the menu item. (The key-character is underlined.) For use with a mouse, however, selection is simplified by the icon button bar. Each button represents a selection available in a pull-down menu. As the mouse pointer is placed over any of the buttons, a description of the button is displayed in the strip along the bottom of the RAPID MDI window.

The following figure shows an example of the RAPID MDI with several of the input windows opened.

7.1 Logon Screen



The first dialog that appears is the Logon. A guest user can logon with a **Username** of "guest" (without the quotes) and a **Password** of "password" (without the quotes). Other **Usernames** can be added through the File | Program Setup | User Table dialog by an administrator.

7.2 Opening Window

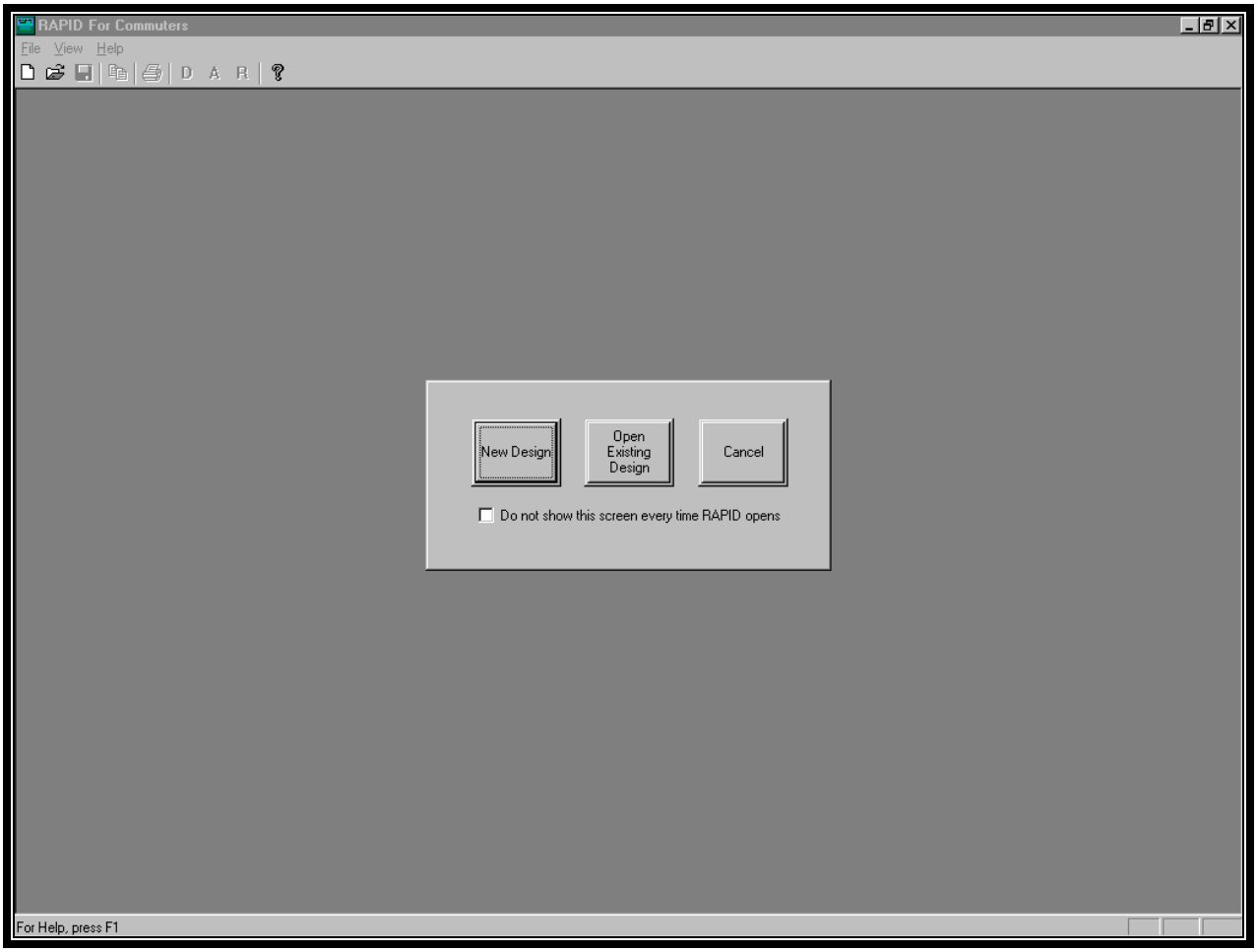


Figure 7-1

After the Logon is complete, the user makes chooses **New Design**, **Open Existing Design**, or **Cancel**.

If the user wants to turn off this screen, check the "Do not show" box. In the future, the user will use the standard Windows **File | New** or **File | Open** menu items.

This feature can be turned back on by going to the Help Menu and clicking on the "Show Opening Screen".

The opening window displays the RAPID Multidocument Interface. All other windows in the RAPID system are selected from and displayed in this window. The **Statusbar** (gray strip along the bottom) describes the function of each button as it is pointed to by the mouse. The **Statusbar** also gives hints for entries throughout the program The **Toolbar** (icon buttons near the top of the window) activates most of the capabilities within RAPID. In every case, their functions are reproduced by selections within the pull-down menus located on the line above the button bar. The icon buttons perform the functions listed below. The bold face entries are documented within this section.

 Starts a **New** repair.

 **Opens** an existing repair file.

 **Saves** the current repair to a file.

 **Copies** the current window to the Windows clipboard.

 **Prints** the current window.

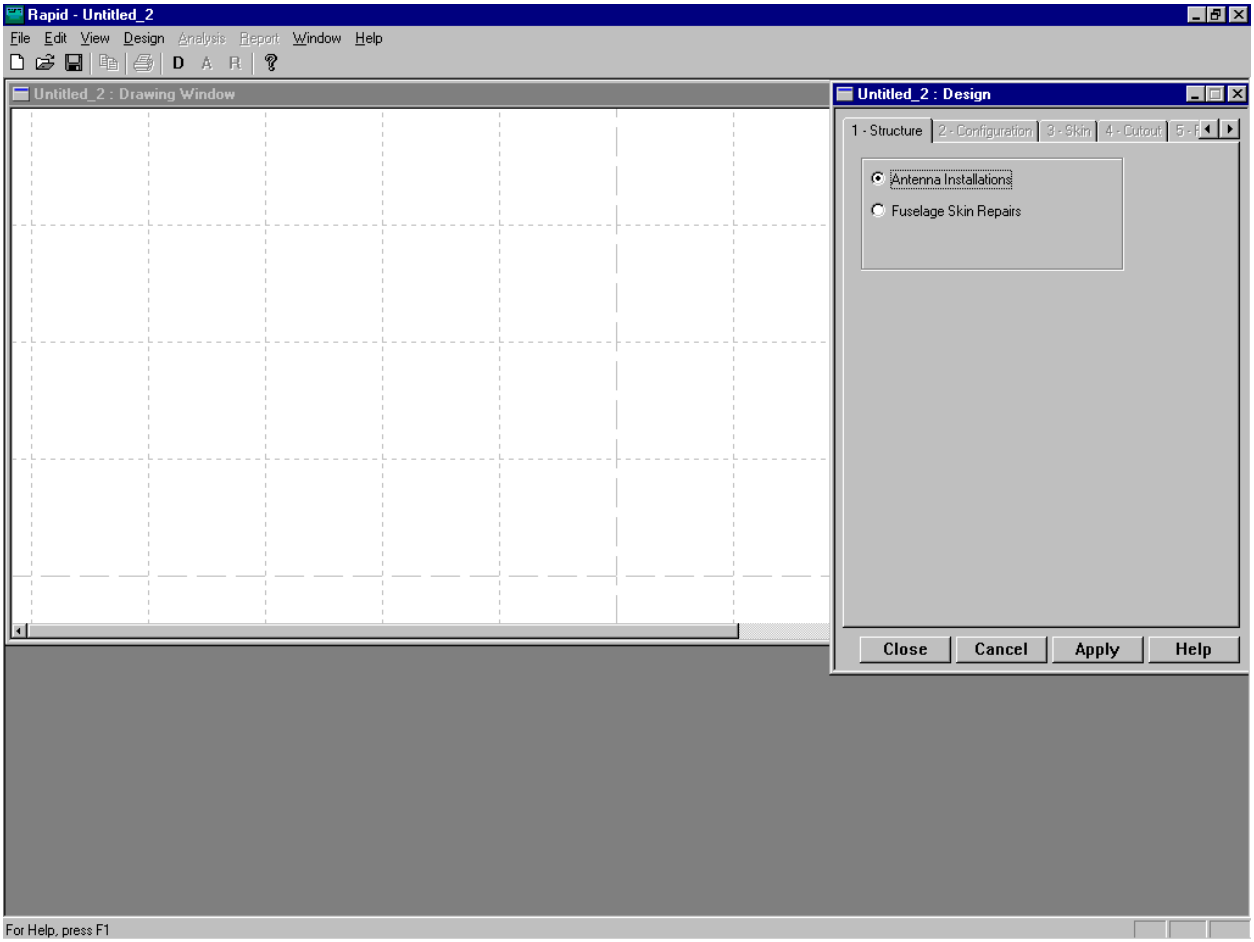
 Activates the **Design** property sheet.

 Activates the **Analysis** property sheet.

 Activates the **Report** property sheet.

 Invokes the on-line **Help** system.

7.3 Opening Design



After choosing either the **New** or **Existing Design**, the user will be presented with a "Design" Property Sheet containing the following pages: **Structure**, **Configuration**, **Skin**, **Cutout**, **Frames**, **Longerons**, **Materials**, and **Geometry**.

After the Structure page is completed and "**Applied**", the next page becomes available.

The pages are disabled to force the user to complete the design in a logical way. As each page is completed and "**Applied**", the next page becomes available. Mouse click on the next page to activate it.

"**Apply**" saves the data in a temporary file. "**Close**" will also save the data; but will close the "Design" Property Sheet as well.

If all the required input is not provided then the program will inform the user what is missing and not activate the next page

NOTE:

As the user progresses through the pages, changes sometimes have to be made on a previous property sheet or pages. If a change is made, the program advises the user which pages must be modified or **Re-Applied**.

7.4 Design Property Sheets

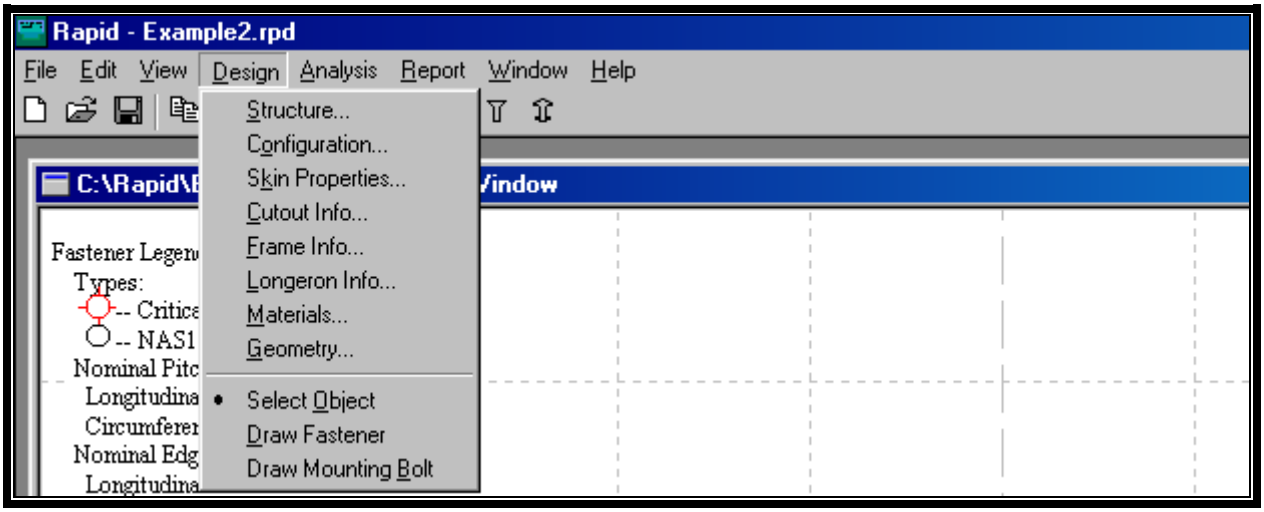


Figure 7-2

This activates the **Design** Property Sheet containing the pages to create a design. On a new design, only **Structure** will be available; the other choices being "grayed out".

On an existing design, the user can use the menu to navigate rapidly to a particular page.

7.4.1 Structure

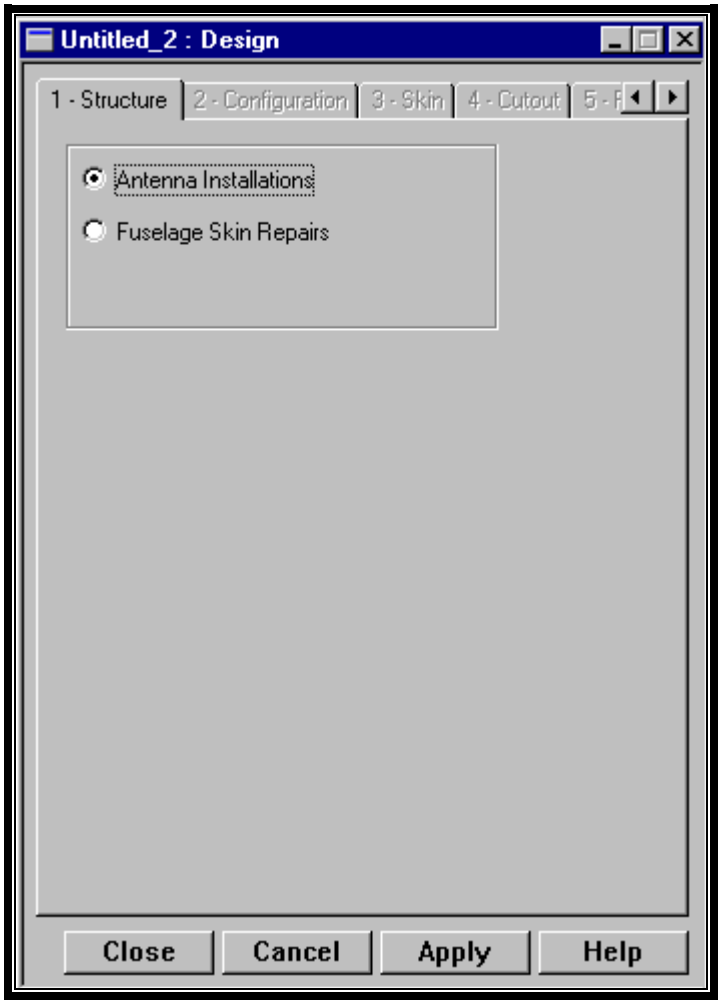


Figure 7-3

The user selects the design type from the Structure page. This tells the program how to format the subsequent pages.

After the Structure page is completed and "**Applied**", the **Configuration** page becomes available.

The pages are disabled to force the user to complete the design in a logical way. As each page is completed and "**Applied**", the next page becomes available. Mouse click on the next page to activate it.

"**Apply**" saves the data in a temporary file. "**Close**" will also save the data; but will close the "Design" Property Sheet as well.

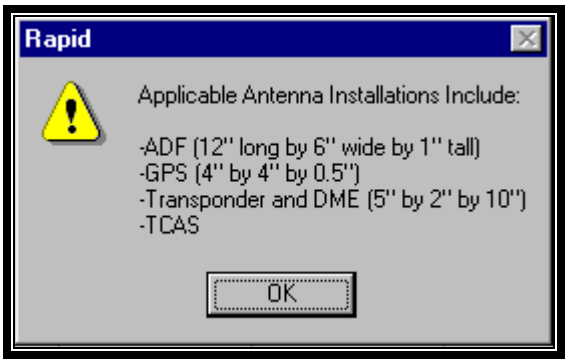


Figure 7-4

If Antenna Installations is selected and applied, the user will be presented with the above message box showing which kind of installations are applicable with this version.

7.4.2 Configuration

There are 2 basic types of Configuration pages. The first type is for Antenna Installations

7.4.2.1 Antenna Installations

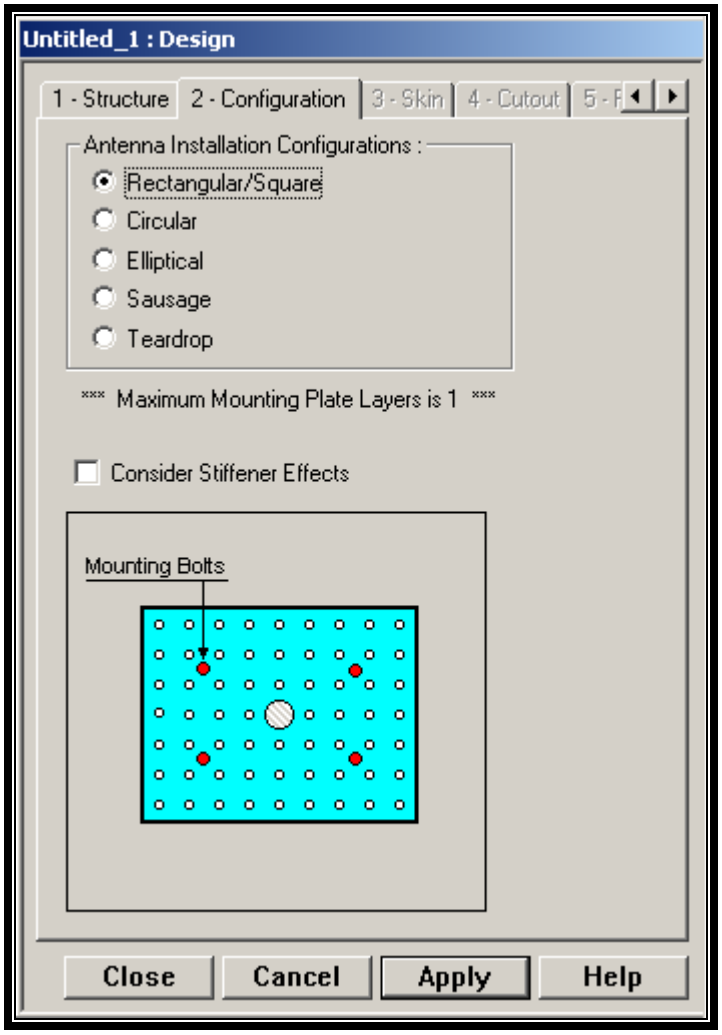


Figure 7-5

The next item in the sequence for Antenna Installations is type of **Configuration** for the design. The user will determine which configuration to use and whether the select **Consider Stiffener Effects**.

After the selections for the configuration are "**Applied**" then the **Skin** page is active.

7.4.2.2 Fuselage Repairs

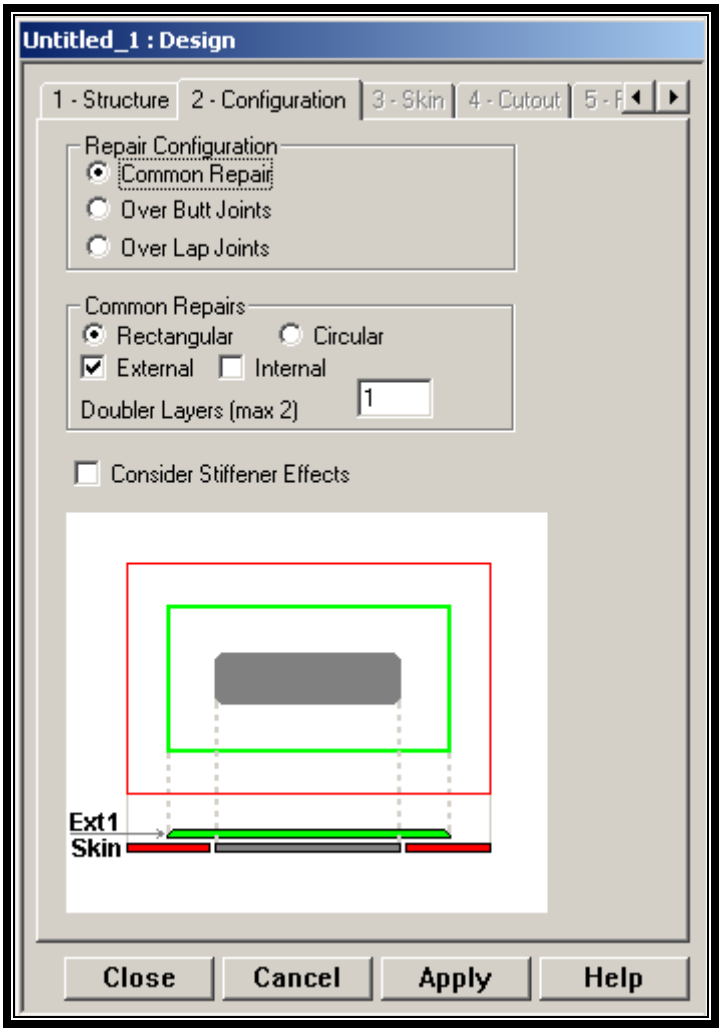
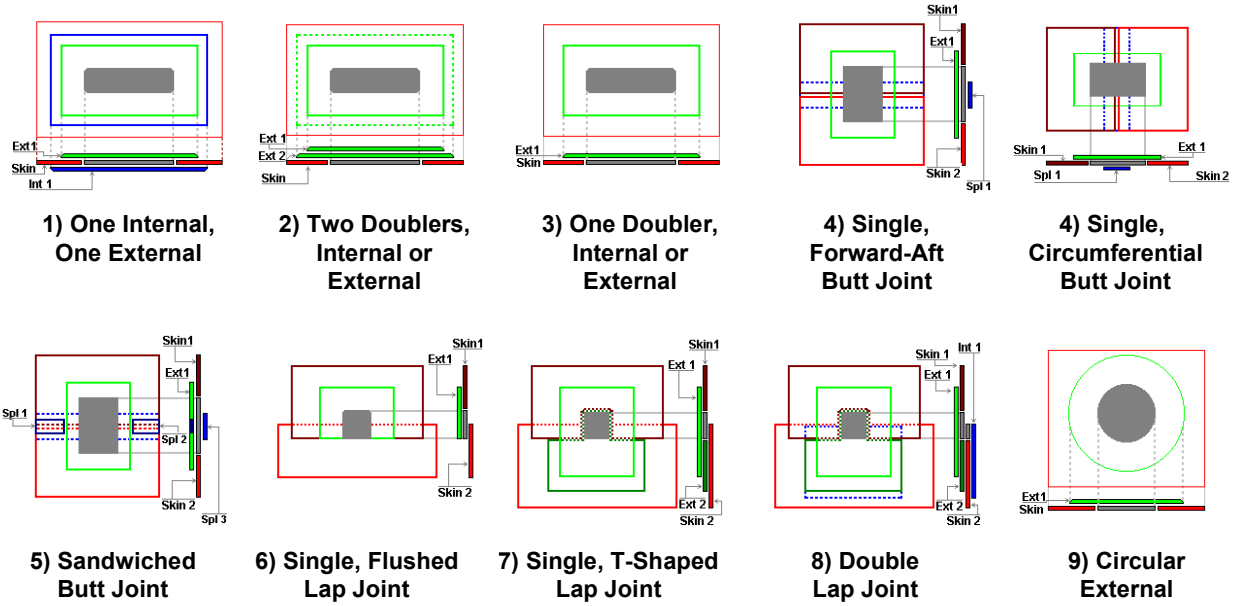


Figure 7-6

The next item in the sequence for Fuselage Repairs is type of **Configuration** for the design.

This window is used to select among the supported fuselage repair types. For the repair type selected, a generic schematic of the cross section is depicted within the window. Currently, nine repair types are available, as depicted in the following figure:



1) One Internal, One External | **2) Two Doublers, Internal or External** | **3) One Doubler, Internal or External** | **4) Single, Forward-Aft Butt Joint** | **4) Single, Circumferential Butt Joint**

5) Sandwiched Butt Joint | **6) Single, Flushed Lap Joint** | **7) Single, T-Shaped Lap Joint** | **8) Double Lap Joint** | **9) Circular External**

The user will determine which configuration to use and whether the select **Consider Stiffener Effects**.

After the selections for the configuration are "**Applied**" then the **Skin** page is active.

7.4.3 Skin

7.4.3.1 Common Repairs and Antenna Installations

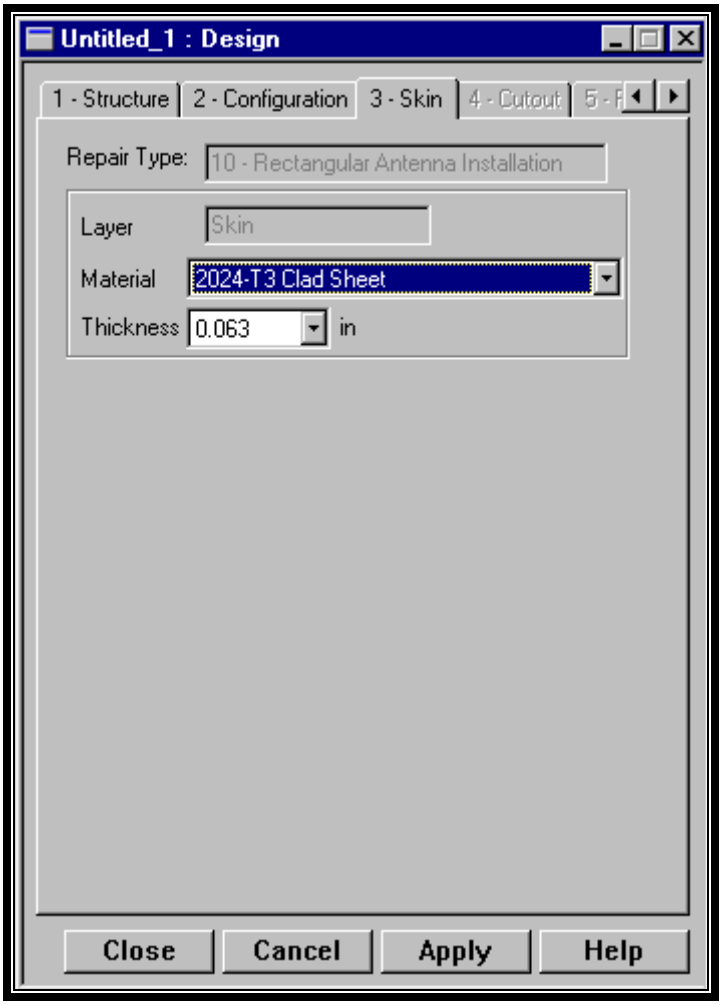


Figure 7-7

The next item in the sequence is the cutout in the skin. Here the dimensions of the cutout are input, then "**Applied**"

After the selections for the configuration are applied then the **Skin** page is active. On this page the material and thickness of the skin is determined

7.4.3.2 Complex Repairs – Butt and Lap Joints

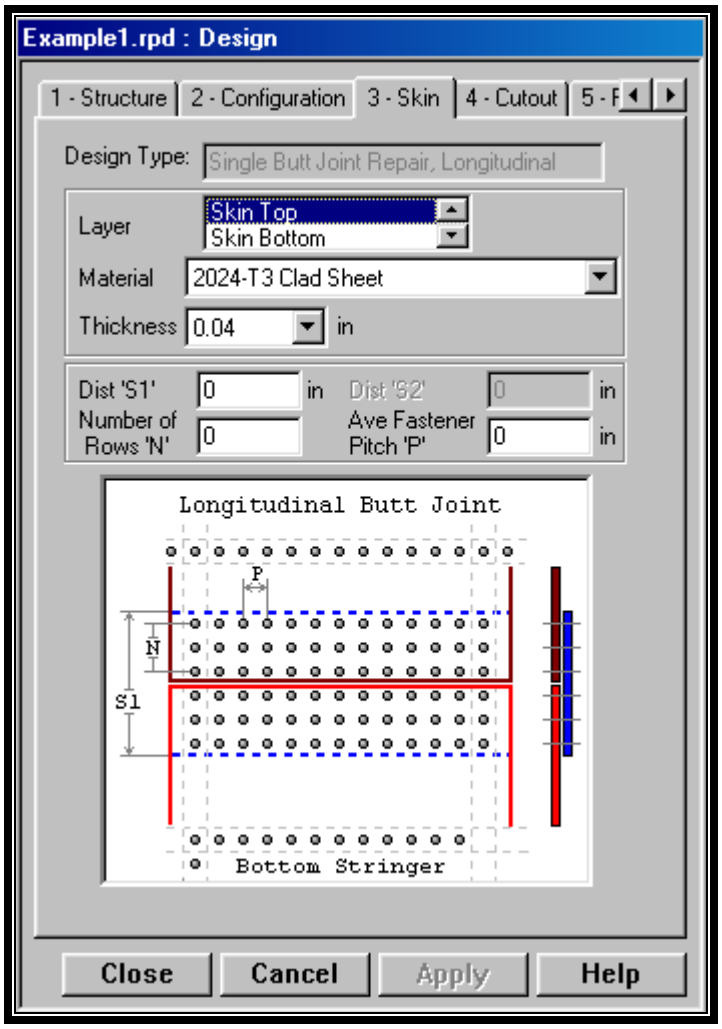


Figure 7-8

This dialog window is used to define the material properties of the skins and joint, and is manually invoked by clicking with the right mouse button on a skin or joint or by selecting the toolbar icon. If the current **Configuration** contains more than one skin or also contains a joint, then the properties for each of those layers must be defined before the window may be closed with the **Close** button.

For repairs over splice and butt joints, the geometry must be entered as indicated in the displayed drawing. This dialog window will display only those fields which are appropriate for the current repair type.

There are different limits as enters are made. The program will advise the user when those limits are violated, and will make suggestions for entries.

The **Close** button updates the current model with any information entered in the window and then closes the window. The **Cancel** button exits the window without saving any new entries. Finally, the **Help** button retrieves help appropriate to this window.

7.4.4 Cutout

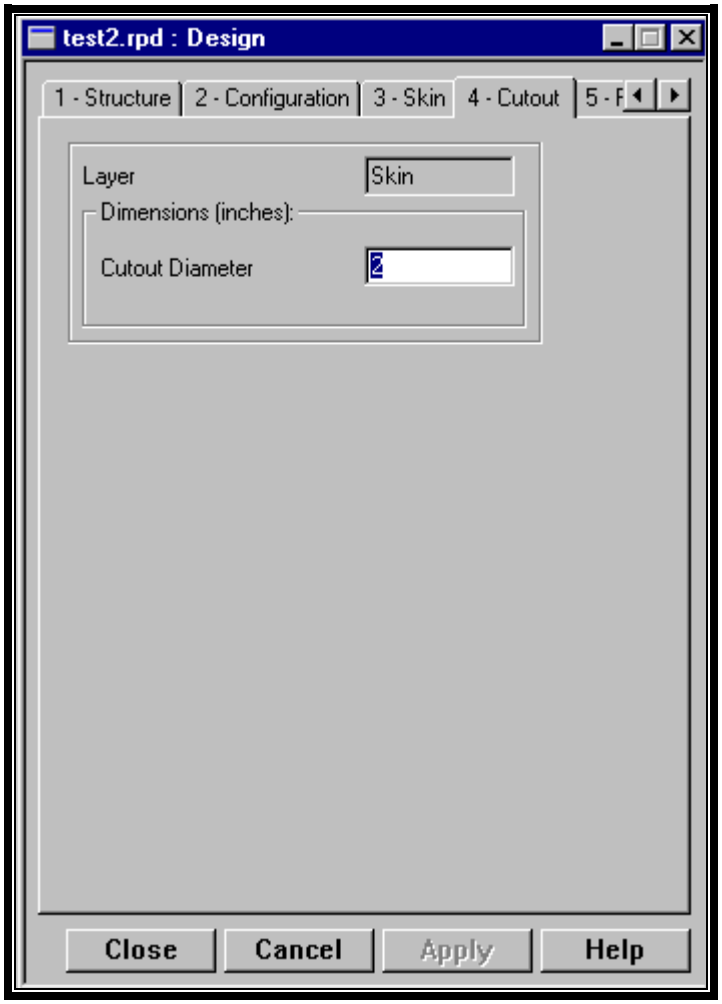


Figure 7-9

The next item in the sequence is the **Cutout** in the skin. Here the dimensions of the cutout are input, then "**Applied**".

In the current version, only a single layer of skin may be cutout, except for the type 8 double lap joint repair which has two overlapping cutouts in the skin lap joint.

There are different limits depending on the type of **Configuration**. The program will advise the user when those limits are violated, and will make suggestions for entries.

The **Close** button updates the current model with any information entered in the window and then closes the window. The **Cancel** button exits the window without saving any new entries. Finally, the **Help** button retrieves help appropriate to this window.

7.4.5 Frames

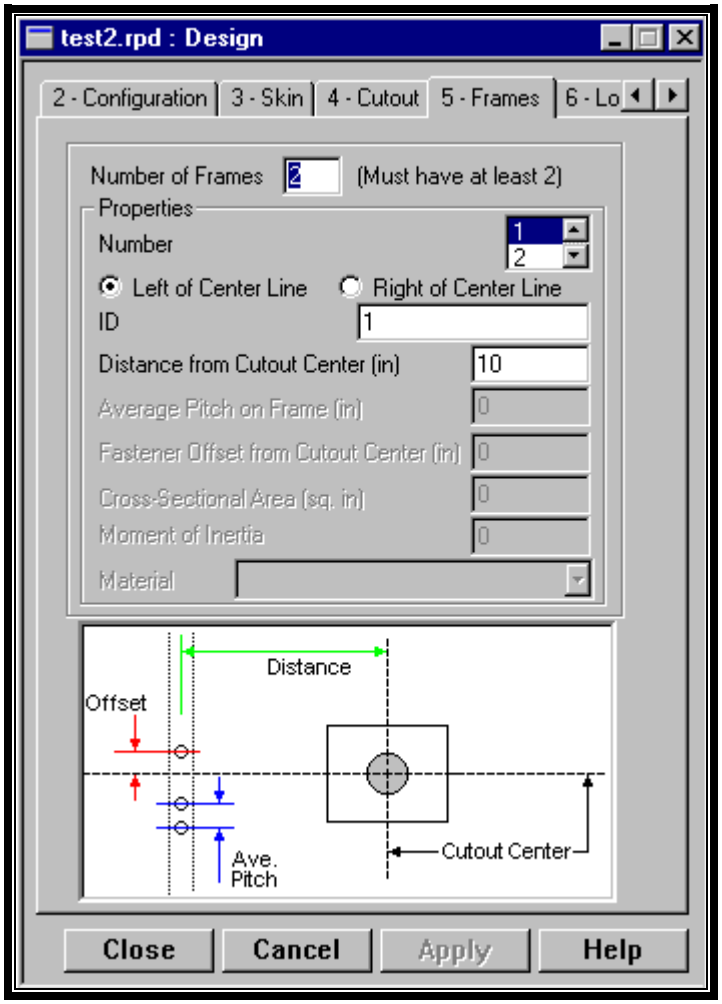


Figure 7-10

The frames (stiffeners) around the installation require definition. This is used to identify the location on the design. A minimum of two frames (stiffeners) is required. If **Consider Stiffener Effects** is checked in the **Configuration** page then the last five fields are active, otherwise they are "grayed" out.

More than 2 frames (stiffeners) can be defined, by increasing the **Number of Frames.**
As the number is increased, the **Number List Box** is automatically changed to reflect the total **Number of Frames**.

The drawing at the bottom of this page indicates where measurements are taken.

Each frame (stiffener) has the following required entries and must be defined:

1. Select the **Number** to be defined.
2. Indicated whether it is above or below the Center Line. If it is on the Center Line; pick either one, as the **Distance** will be set to 0.
3. Enter an **ID** for the frame; usually identified by using the aircraft schematic location.
4. Enter the **Distance** from Center Line. Limits are 0.0 to 100.0 inches.

If **Consider Stiffener Effects** is checked on the **Configuration** page, the following are required entries:

1. **Average Pitch on Frame**. Limits are 0.5 to 2.0 inches.
2. **Fastener Offset from** Center Line. Limits are 0.0 to 1.5 inches.
3. **Cross-Sectional Area**. Limits are 0.1 to 2.0 in^2
4. Select the Frame (stiffener) **Material**

Repeat the process for each frame (stiffener).

7.4.6 Longerons

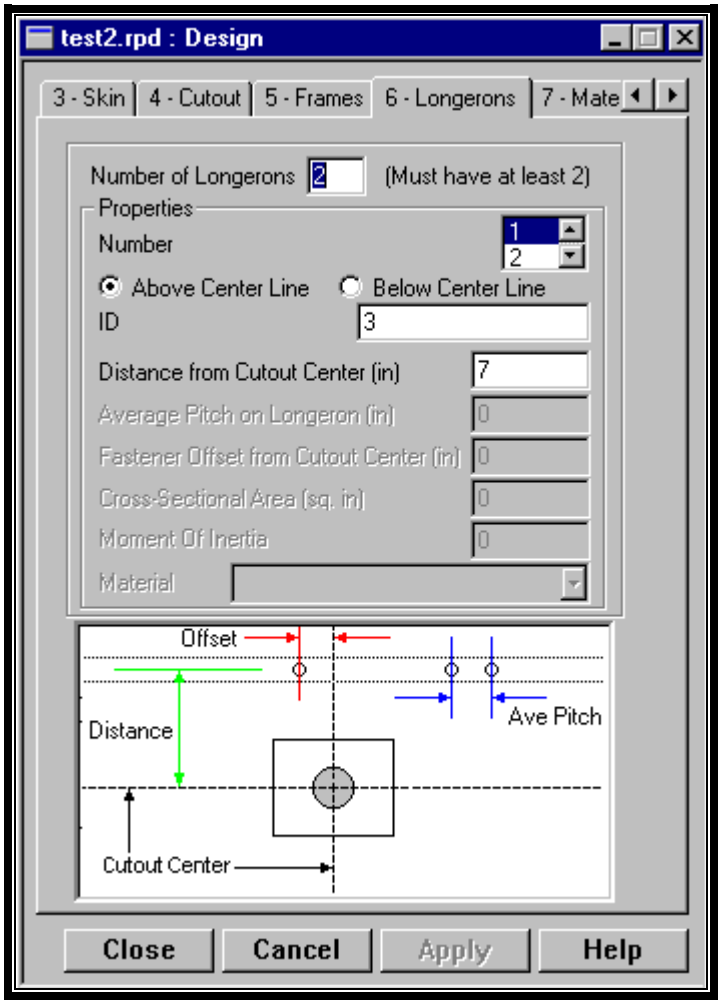


Figure 7-11

The longerons around the installation require definition. This is used to identify the location on the design. A minimum of two longerons is required. If **Consider Stiffener Effects** is checked in the **Configuration** page then the last five fields are active, otherwise they are "grayed" out.

More than 2 longerons can be defined, by increasing the **Number of Longerons.**
As the number is increased, the **Number List Box** is automatically changed to reflect the total **Number of Longerons**.

The drawing at the bottom of this page indicates where measurements are taken.

Each longeron has the following required entries and must be defined:

1. Select the **Number** to be defined.
2. Indicated whether it is above or below the Center Line. If it is on the Center Line; pick either one, as the **Distance** will be set to 0.
3. Enter an **ID** for the longeron; usually identified by using the aircraft schematic location.
4. Enter the **Distance** from Center Line. Limits are 0.0 to 50.0 inches.

If **Consider Stiffener Effects** is checked on the **Configuration** page, the following are required entries:

1. **Average Pitch on Longeron**. Limits are 0.5 to 2.0 inches.
2. **Fastener Offset from** Center Line. Limits are 0.0 to 1.5 inches.
3. **Cross-Sectional Area**. Limits are 0.1 to 2.0 in^2
4. Select the Frame (stiffener) **Material**

Repeat the process for each longeron.

7.4.7 Material

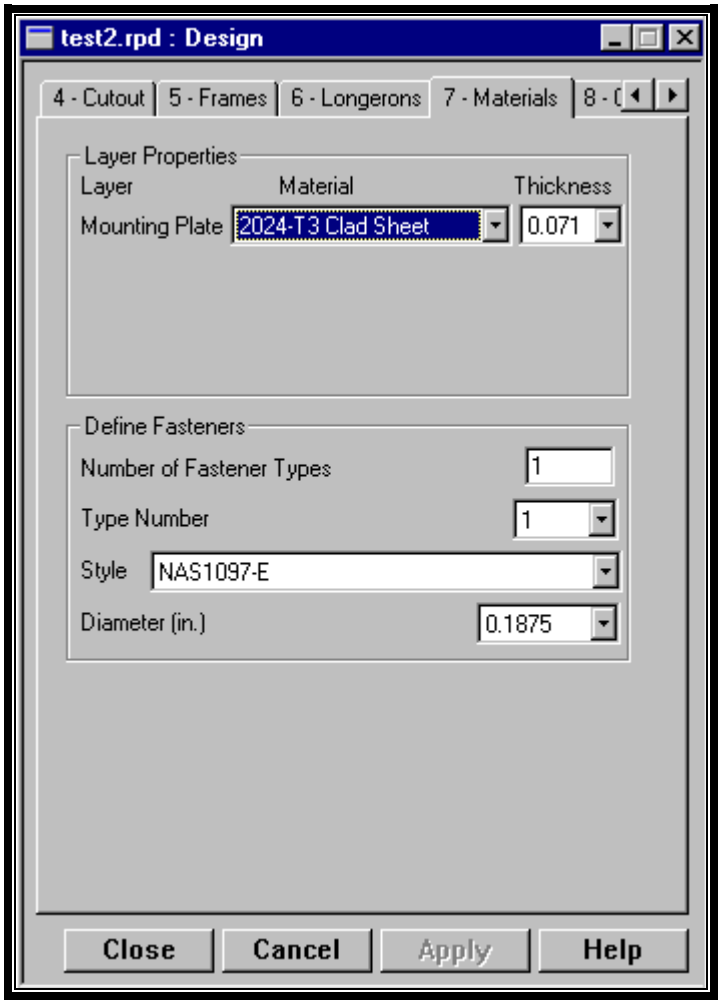


Figure 7-12

The **Materials** page describes the **Layer Properties** for each layer of the design and **Defines Fasteners** used in the design

7.4.7.1 Layer Properties

The **Layer Properties** lists all the layers in the design except the **Skin** layer.

Select the appropriate **Material** and **Thickness** for each **Layer**.

7.4.7.2 Define Fasteners

The user specifies the different types and styles of fasteners in the design using the **Define Fasteners** dialog. Fastener types are defined by first entering the number of fastener types desired in the **Number of Fastener Types** edit field. The **Type Number** combobox will automatically be updated to reflect the **Number of Fastener Types**.

Style and **Diameter** must be entered for each **Type Number**:

1. Select a **Type Number**.
2. Select a **Style** to be associated with the **Type Number**.
3. Select a **Diameter** for the **Style** selected.
4. Repeat the process for each **Type Number**.

7.4.7.3 Change Fastener Types on the Drawing

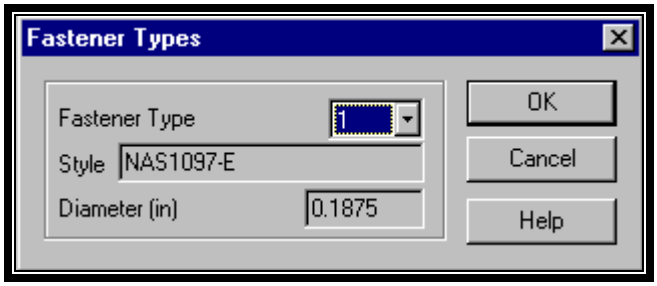


Figure 7-13

To change fastener types, they must first be defined in the **Define Fasteners** on the **Material** page. Any number of fasteners may be simultaneously changed to a predefined type. Multiple fasteners are selected by click and holding the left mouse button; then dragging the mouse to draw a rectangle around a group of fasteners. Releasing the mouse button then toggles the selection state of each of the enclosed fasteners. Subsequently clicking the right mouse button will invoke the **Fastener Types** dialog through which the new fastener type may be selected.

7.4.8 Geometry

There are 2 types of Geometry pages. The first type is for Antenna Installations

7.4.8.1 Antenna Installations

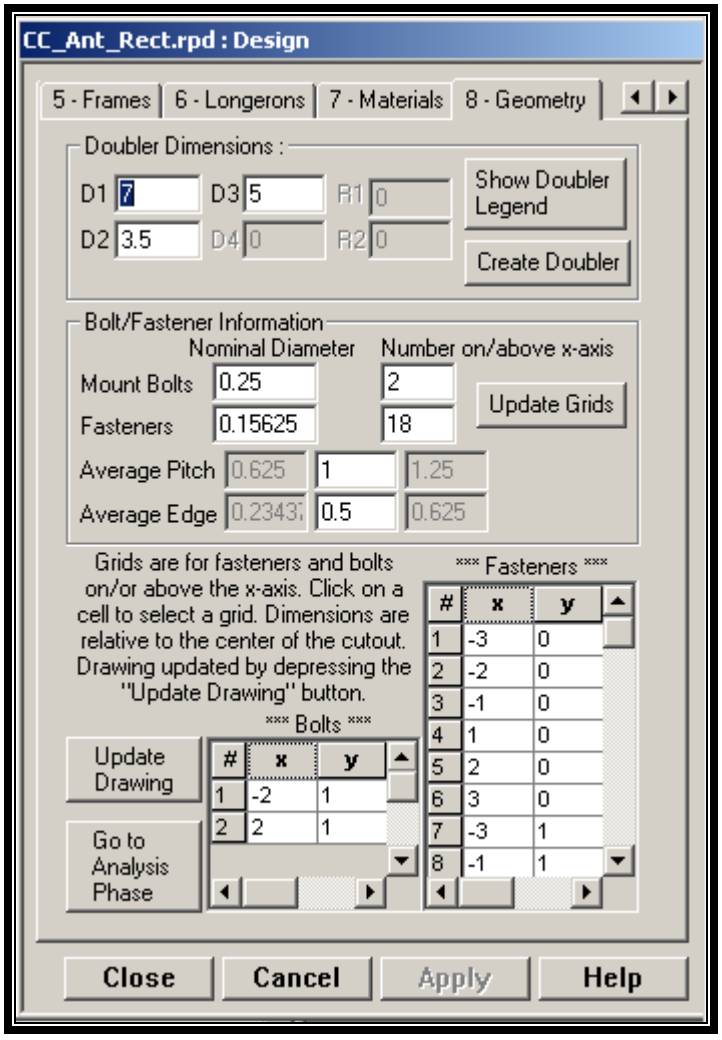


Figure 7-14

The Geometry page provides the input for the Doubler Dimensions, Bolt and Fastener information, and the exact locations of the bolts and fasteners. For antenna installation designs, the user is allowed to locate fastener anywhere within the mounting plate (doubler). All antenna designs must be symmetrical about the x-axis.

You must enter the doubler dimensions before the rest of the dialog becomes available. Depress **Show Doubler Legend** button for description of the dimension entries. After entering the dimensions; depress the **Create Doubler** button. The drawing window will display the mounting (doubler) you have defined.

Next, enter the bolt and fastener information. The nominal fastener diameters can be changed, but must be within the range of the types of fasteners chosen on the **Materials** page. As designs MUST be symmetrical about the x-

axis, the user only enters the number of bolts and fasteners above the x-axis. After entering the average pitch, depress the **Update Grids** button. This will populate the Bolt and Fastener grid with the appropriate number of cells. Enter the exact locations (distance from the center if the cutout) in both the x and y directions. Once all the bolts and fasteners are entered, depress either the **Apply** or the **Update Drawing** buttons. This will display the bolts and fasteners in the Drawing Window.

Fastener and bolts can be updated by entering new values in the appropriate grid cell. They can also be updated by Selecting a fastener (bolt) above the x-axis; and either moving it with the mouse or the cursor keys. The corresponding fastener (bolt) below the x-axis will also be moved.

Click the "**Close**" button to close the Design property sheet.

Mouse click the "A" on the tool bar to activate the Analysis property sheet.

Notes:

The **Nominal Diameter** defaults to the Type 1 fastener defined in the Material page. The **Nominal Diameter** is used to established the Pitch and Edge min/max values from the Advisory System.

Pitch and Edge Distance:

Under the **Pitch and Edge Distance (in)** section of the dialog window, the user specifies the pitch and edge distances of the fasteners, which otherwise default to the settings selected in the program setup. These values are bound by the min/max values stored in the Advisory System as determined by the nominal fastener diameter. In models with stiffener effects and/or joints with existing fasteners, the pitch is fixed to that of the existing fasteners.

7.4.8.2 Fuselage Repairs

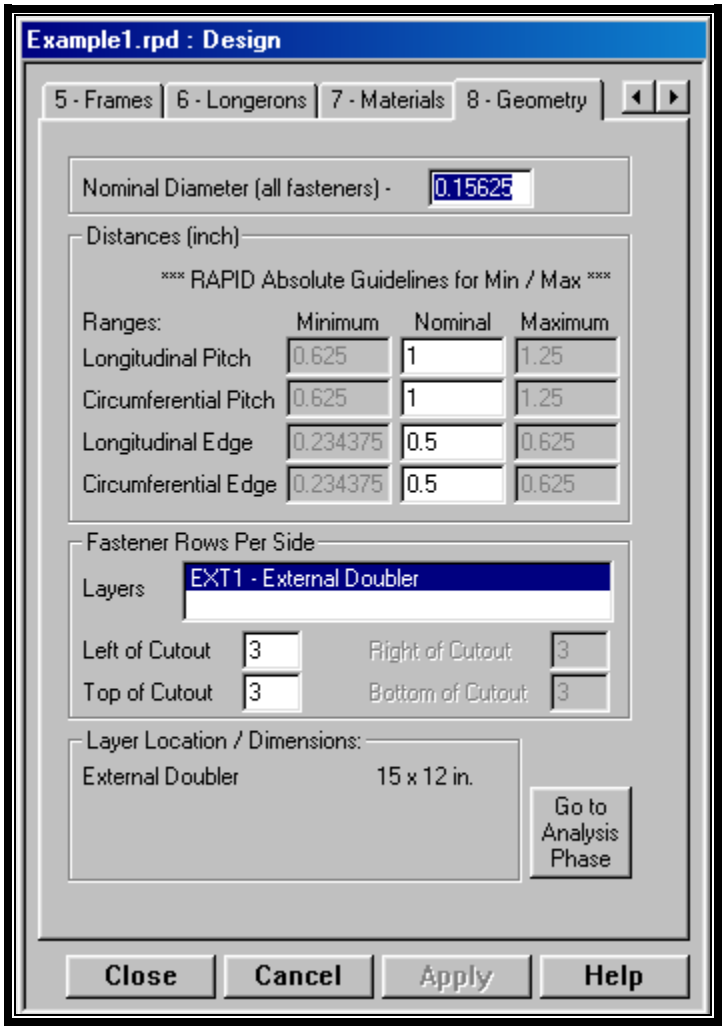


Figure 7-15

The Geometry page provides the input for the Nominal Pitch, Pitch and Edge Distances, Fastener Rows per Side. The dimensions of the design layers are displayed near the bottom of the page.

After "**Applying**" the geometry, mouse click the "**Close**" button to close the Design property sheet.

Mouse click the "A" on the tool bar to activate the Analysis property sheet.

The **Nominal Diameter** defaults to the Type 1 fastener defined in the Material page. The **Nominal Diameter** is used to established the Pitch and Edge min/max values from the Advisory System.

Pitch and Edge Distance:

Under the **Pitch and Edge Distance (in)** section of the dialog window, the user specifies the pitch and edge distances of the fasteners, which otherwise default to the settings selected in the program setup. These values are

bound by the min/max values stored in the Advisory System as determined by the nominal fastener diameter. In models with stiffener effects and/or joints with existing fasteners, the pitch is fixed to that of the existing fasteners.

Fastener Rows Per Side:

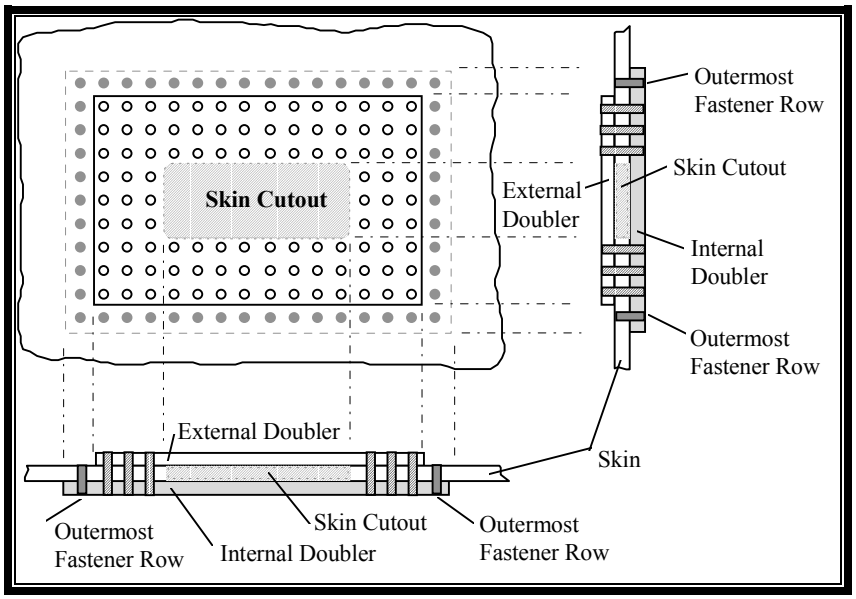
Each layer, other than the **Skin**, must have fastener rows defined:

1. **Select a Layer.**
2. Enter the appropriate number of rows for each side of the cutout. Some of the edit fields may not be accessible. This could be that the design does not call for a row of fasteners or the design is symmetrical; in which case, the program will automatically enter the rows necessary. Use the guidelines shown in the figures below. Rule of thumb: For 2 layers on the same side of the skin, the layer closest to the skin will always have an extra row of fasteners. An internal layer will always have a extra rows of fasteners, if used with an external layer.
3. Repeat the process for each layer.

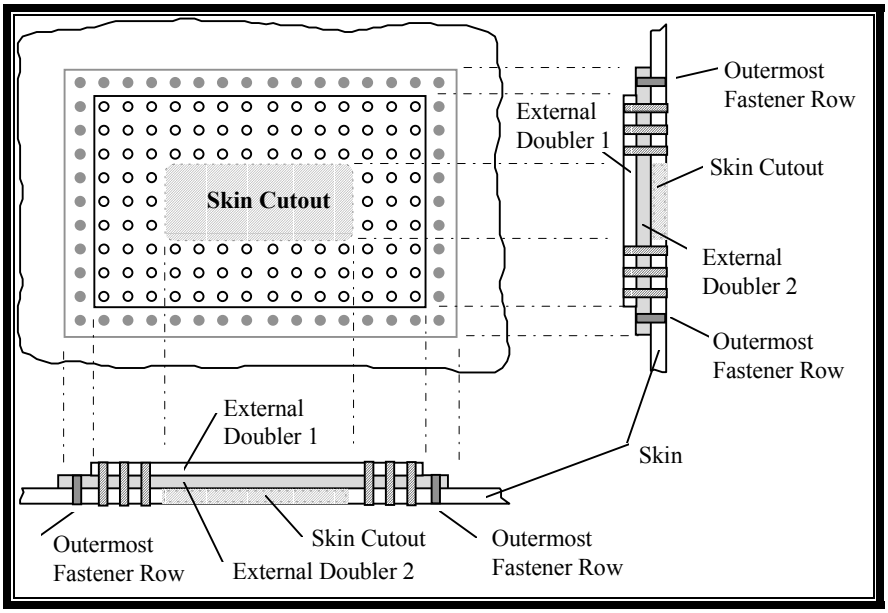
Fastener Row limits vary by type and configuration. Limits are 1 to 5; the program will give hints on the status bar.

Fastener Row Examples:

For repair type 1, the internal doubler should be larger than the external doubler. The internal doubler should be sized such that the single outermost fastener row passes through the skin and the internal doubler as shown in the figure below.



For repair type 2, the external doubler 2 (inner) should be larger than the external doubler 1 (outer). The doublers should be sized such that the single outermost fastener row passes through the skin and the external doubler 2 (inner) as shown below.



7.4.8.3 Joint Allowable Calculations (both types):

"Outer" layer allowables are contained in the Fastener Database. "Inner" layer allowables (straight shank – single/double shear) are derived by calculating the shear allowable and the bearing allowable for the fastener at each layer and then using the smaller of the two as the joint allowable. The following formulas were used in the calculations:

Countersunk Fasteners with Straight Shank Holes:

Shear Allowable = Shear Strength(lbs) x Correction Factor x 2 (if Double Shear)

Bearing Allowable = F_{BRU} (Layer Material) x Layer Thickness x Fastener Hole Diameter x 1000

Straight Shank Fasteners:

Shear Allowable = F_{SU} (Fastener) x Shank Area x Correction Factor x 2 (if Double Shear) x 1000

Bearing Allowable = F_{BRU}(Layer Material) x Layer Thickness x Fastener Hole Diameter x 1000

1. Shear Strength from Fastener Database (data from MIL-HDBK-5F, 1 November 1990)
2. Correction Factor from MIL-HDBK-5F, 1 November 1990, Table 8.1.2.1(b)
3. F_{BRU} from Material Database (e/D = 2.0 from MIL-HDBK-5F, 1 November 1990)
4. F_{SU} from Fastener Database ((data from MIL-HDBK-5F, 1 November 1990)
5. Shank Area = ((Fastener Diameter)^2 x PI) / 4

7.5 Analysis Property Sheet

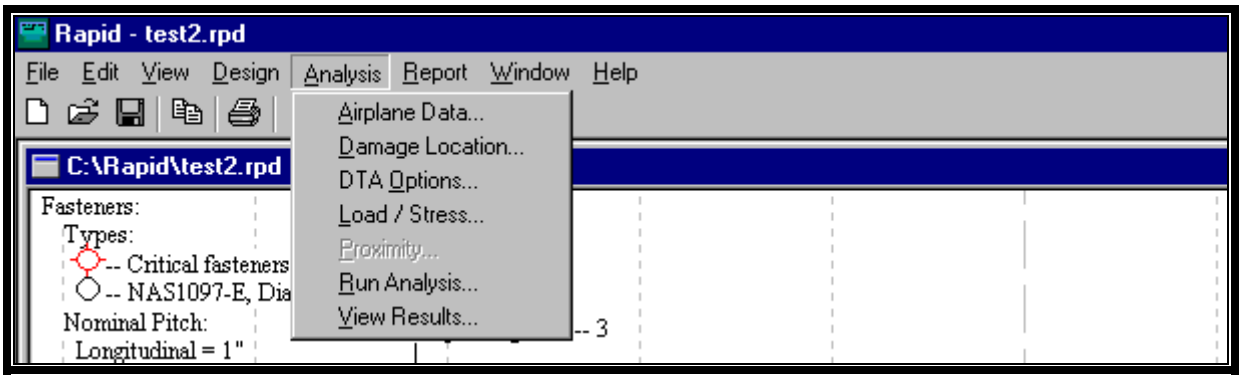


Figure 7-16

This activates the **Analysis** Property Sheet containing the pages to create a design. On a new design, only **Airplane Data** will be available; the other choices being "grayed out".

On an existing design, the user can use the menu to navigate rapidly to a particular page. In this example, the existing design did not call for **Consider Proximity Effects** in the **DTA Options page**, so the **Proximity** page remains unavailable

7.5.1 Airplane Data

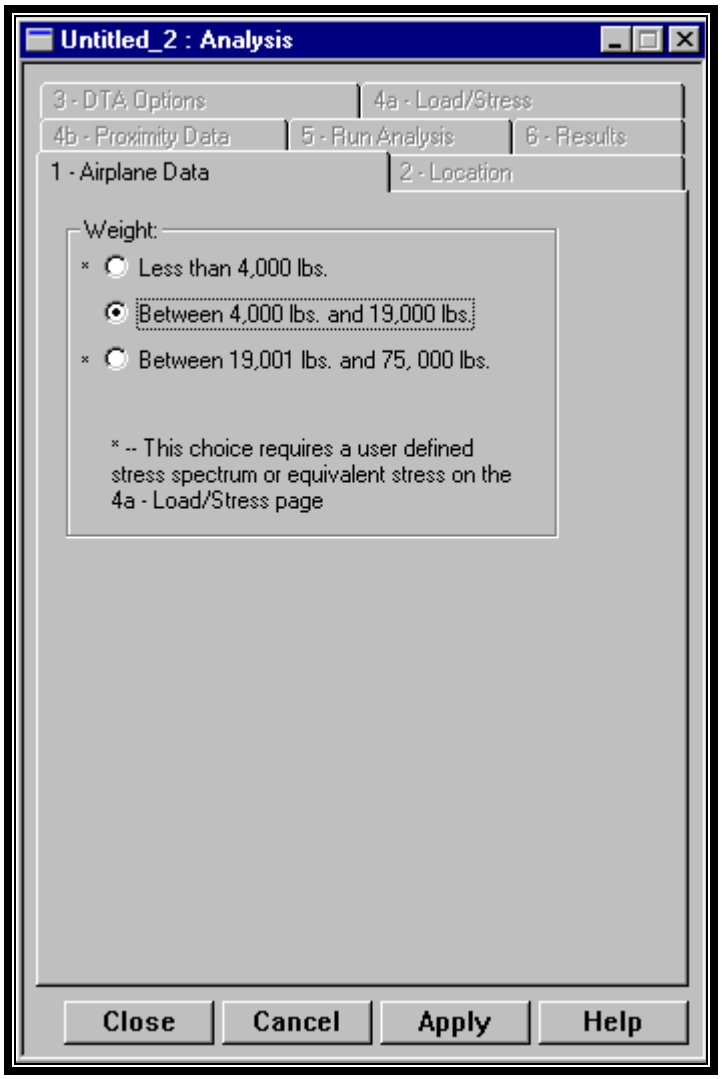


Figure 7-17

This is the first page of the "**Analysis**" property sheet. The pages contain are **Airplane Data**, **Location**, **DTA Options**, **Load/Stress**, **Proximity Data**, **Run Analysis**, and **Results**.

The **Airplane Data** is used to determine the type of airplane and whether RAPID can supply spectrum data for the **Load/Stress** page to use to calculate equivalent stress.

If RAPID can't supply the spectrum data, the user is advised to provide a stress spectrum or equivalent stress.

7.5.2 Location

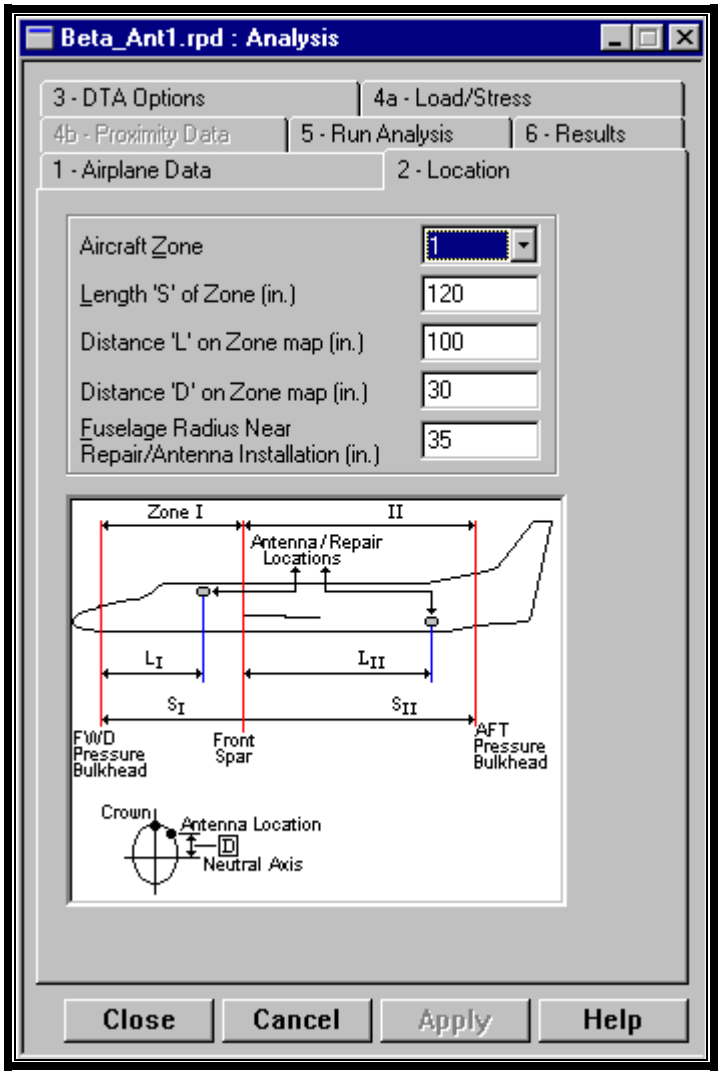


Figure 7-18

The **Location** page is where the user defines where the design location resides on the fuselage.

This window is used to define the location of the damage on the aircraft in relation to several pre-defined regions, which may be viewed by drawing at the bottom of the page.

This input is used to when calculating the equivalent stress at the design location.

1. Select the **Zone** in which the design location resides.
2. Enter the total distance for the **Zone** in **Length "S" of Zone**
3. Enter the distance from the front of the **Zone** to the design location in **Distance "L" on Zone map**
4. Enter the distance from the **Distance "D" on Zone map**
5. Enter the **Fuselage Radius Near Design Location**

The **OK** button updates the current model with any information entered in the window and then closes the window. The **Cancel** button exits the window without saving any new entries. Finally, the **Help** button retrieves help appropriate to this window.

7.5.3 DTA Options

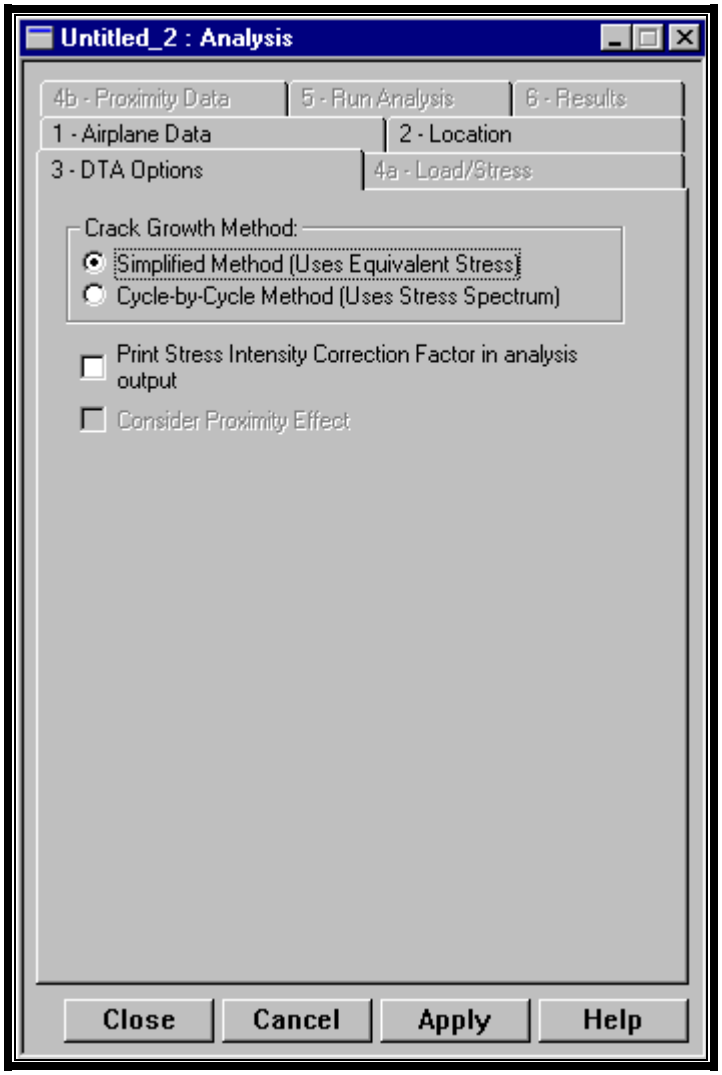


Figure 7-19

The **DTA Options** page provides options for the damage tolerance analysis.

It is recommended to use the Simplified Method (equivalent stress method) initially instead of the cycle by cycle method. The cycle by cycle method will take **hours** to run. The equivalent stress method will provide answers approximate to the cycle by cycle method but much faster. If the equivalent stress results are not adequate then the user should use the cycle by cycle method.

Proximity effects are currently only available for **Common Fuselage Repairs**

7.5.4 Load/Stress

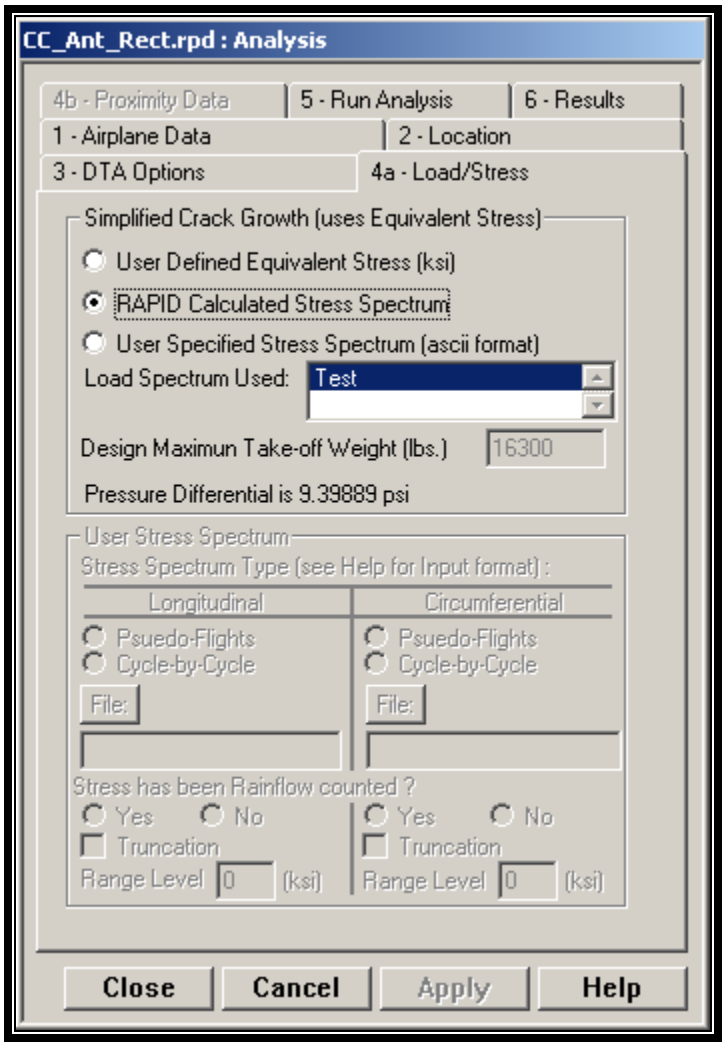


Figure 7-20

The **Pressure Differential** is used to define the limit load condition. User input is required if **User Defined Equivalent Stress** or **User Defined Stress Spectrum** is selected. See section 3 and the Analysis Methods Document for further details on the limit load condition

The **Load/Stress** page is where the load and stress information is input.

After "Applying" the input, the program may run a fortran dll to calculate the equivalent stress. The message bar at the bottom of the program window will indicate if this is happening. The fortran dll only runs for a minute or two.

This dialog window is used to select the stress spectrum applied during the damage tolerance analysis. The **Equivalent Stress** defines the constant stress amplitude at the repair location. See section 3 and the Analysis Methods Document for further details on the equivalent stress. If the equivalent stress is supplied, then the other input fields are not used. Otherwise, the **Stress Spectrum** is selected from one of the **RAPID Defined** spectra or from a **User Defined** file.

User Defined Stress Spectrum (ascii) must be in one of the two following ASCII text file formats. Examples of these files are included with the RAPID distribution as `PSEUDO.SSF` and `CYCLES.SSF` for pseudo-flights and cycle-by-cycle, respectively. These are only examples of the input format; they are not intended to be used as test files.

1. **Pseudo-Flights (in repeatable flight sequence)**

   mtype
   mflt
   $ifltseq_1$ $ifltseq_2$ $ifltseq_3$ … $ifltseq_{mflt}$
   $mcyc_1$
   $Smin_1$ $Smax_1$ $Smin_2$ $Smax_2$ … $Smin_{mcyc1}$ $Smax_{mcyc1}$
   $mcyc_2$
   $Smin_1$ $Smax_1$ $Smin_2$ $Smax_2$ … $Smin_{mcyc1}$ $Smax_{mcyc2}$
   ...
   $mcyc_{mtype}$
   $Smin_1$ $Smax_1$ $Smin_2$ $Smax_2$ … $Smin_{mcyc1}$ $Smax_{mcycmtype}$

   where mtype is the number of different flight types specified (maximum=20)
   mflt is the number of flights in the spectrum (maximum=6000)
   ifltseq contains the sequence of the flight types
   mcyc is the number of cycle pairs for the flight type (maximum=1000)
   S_{min} is the minimum stress
   S_{max} is the maximum stress

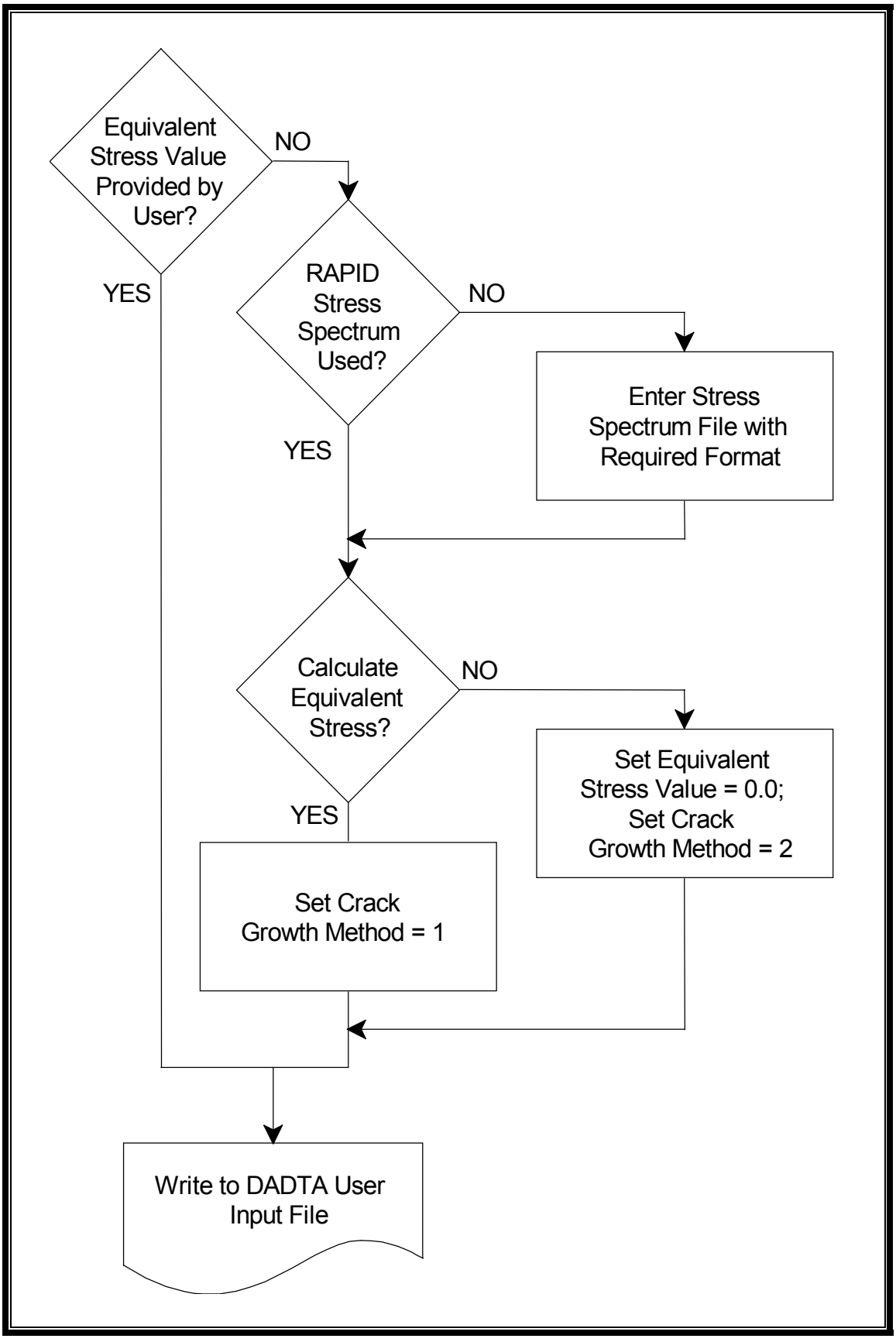
2. **Cycle-by-Cycle Stresses (in repeatable block)**

   mflt
   $mcyc_1$
   $Smin_1$ $Smax_1$ $Smin_2$ $Smax_2$ … $Smin_{mcyc1}$ $Smax_{mcyc1}$
   $mcyc_2$
   $Smin_1$ $Smax_1$ $Smin_2$ $Smax_2$ … $Smin_{mcyc1}$ $Smax_{mcyc2}$
   ...
   $mcyc_{mtype}$
   $Smin_1$ $Smax_1$ $Smin_2$ $Smax_2$ … $Smin_{mcyc1}$ $Smax_{mcycmtype}$

   where mflt is the number of flights in the spectrum (maximum=6000)
   mcyc is the number of cycle pairs for the flight type (maximum=1000)
   S_{min} is the minimum stress
   S_{max} is the maximum stress

For the user defined spectrum, **Rainflow Counting** and **Range Truncation Level** must also be specified, as documented in the Analysis Methods Document.

The input is parsed by RAPID according to the following flow chart:


Equivalent Stress Value Provided by User?
NO
YES
RAPID Stress Spectrum Used?
NO
YES
Enter Stress Spectrum File with Required Format
Calculate Equivalent Stress?
NO
YES
Set Equivalent Stress Value = 0.0; Set Crack Growth Method = 2
Set Crack Growth Method = 1
Write to DADTA User Input File

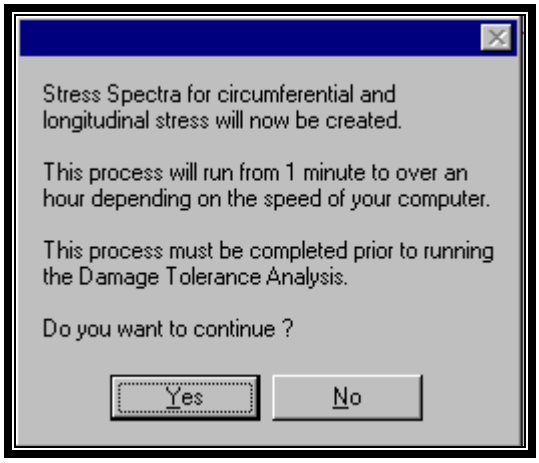


Figure 7-21

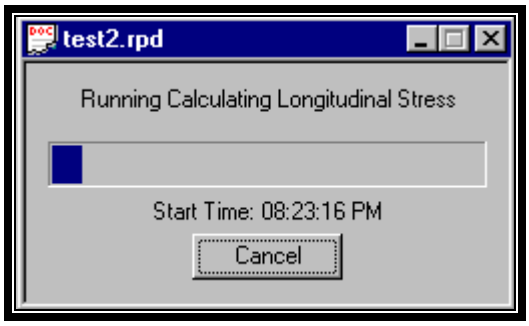


Figure 7-22

7.5.5 Proximity Data

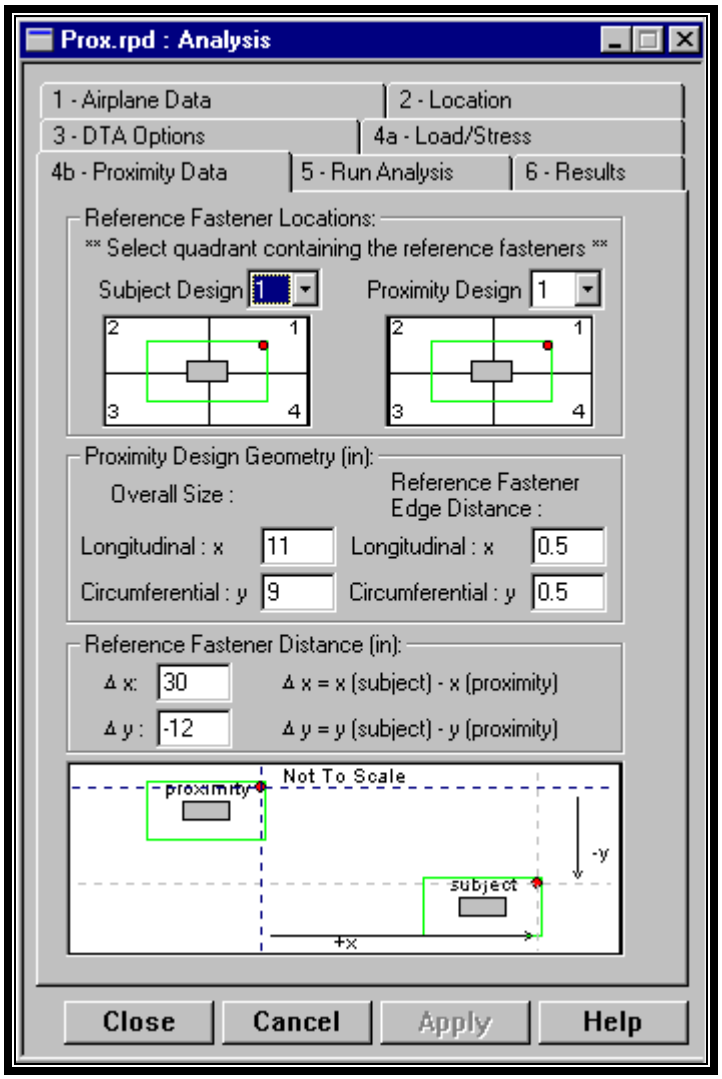


Figure 7-23

This page is for input of the proximity data and is only available if **Consider Proximity Effects** was checked on the **DTA Options** page. **Proximity Effects** are currently only available for **Common Fuselage Repairs**

The **Proximity Data** is used to define the size and location of a nearby repair so that its influence may be considered in the analysis.

The location of the nearby (proximity) design is indicated by selecting a reference fasteners in both the subject and proximity designs in the **Reference Fastener Locations** box.. This is done by selecting the quadrant where the reference fastener is located. You can use the quadrant combo boxes or you can mouse click in the quadrant on the pictures. The picture at the bottom of the dialog page will update as you make you selections, helping you to visualize the relationship.

The geometry of the nearby (proximity) design can be entered directly under the **Proximity Design Geometry** section. Enter the overall size of the proximity repair and the reference fastener edge distance for the proximity design.

The **Reference Fastener Distances** between the Subject reference fastener and the Proximity reference fastener are entered in the **Δx** and **Δy** using the given formulas. Always remember that these distances are really the relationship of the proximity repair to the subject repair. Negative numbers are permitted as indicated in the picture at the bottom of the dialog page. Positive and negative numbers will change the picture accordingly.

7.5.6 Run Analysis

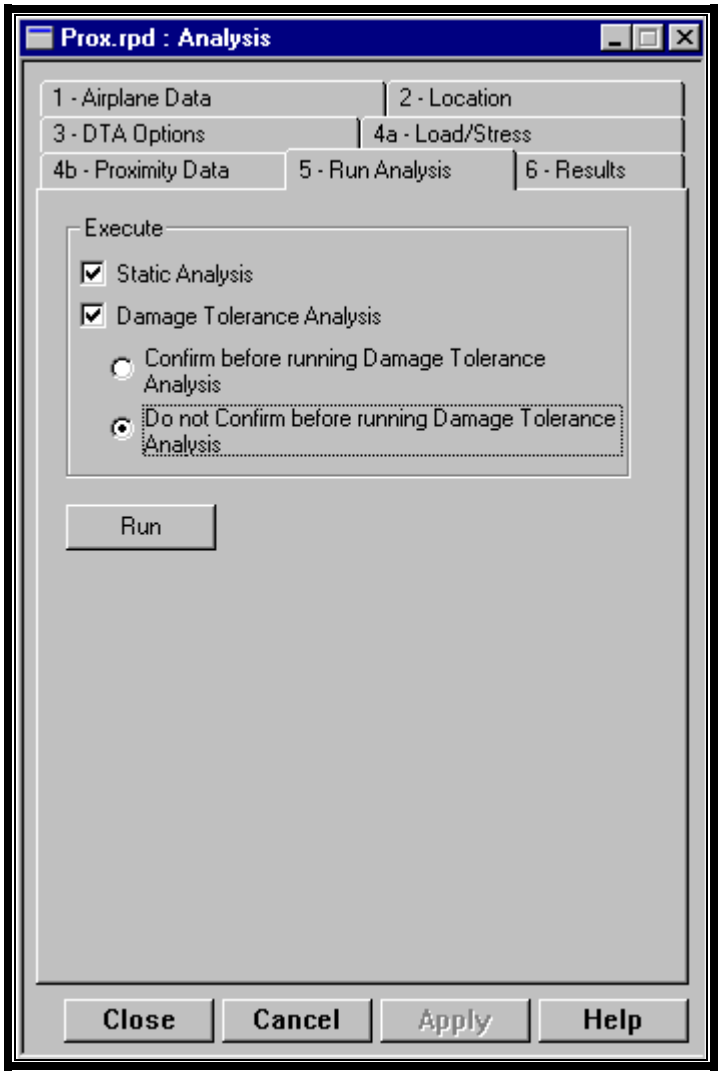


Figure 7-24

On this **Run Analysis** page, the user can review the analysis input files.

The user decides which Analyses to run by checking the appropriate boxes, and indicating whether to be notified after the **Static Analysis** has completed. The **Run** button becomes available after checking a box(es). Depressing the **Run** button starts the Fortran DLLs. The dialog box below opens to indicate which analysis is running and when it started. Once the analysis finishes, the elapsed time is recorded under the **Run** button.

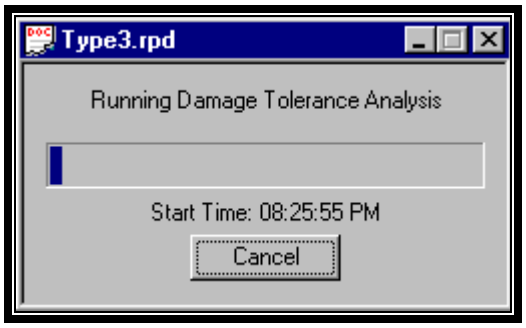


Figure 7-25

This dialog box indicates to the user that the analysis is running and the time it started

The **Cancel** button stops the analysis. **HOWEVER**, the user will be advised that computer memory is probably corrupted and should reboot the computer.

The source of this problem is still being investigated; use the **Cancel** button only as a last resort.

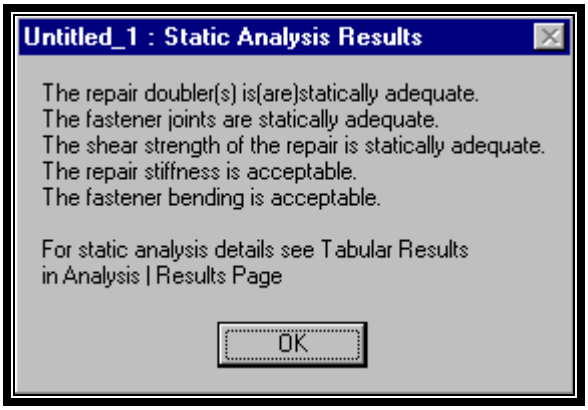


Figure 7-26

The output can be displayed by selecting the **Tabular Results** on the **Results** page.

From the static analysis, the margins of safety (MS) based on the repair doubler allowable and the fastener joint allowables are calculated to determine the adequacy of the repair as follows.

MS < 0: Repair is statically inadequate
MS = 0: Repair is marginally adequate
MS > 0: Repair is statically adequate

For repairs that are statically inadequate and marginally adequate, the repair should be redesigned.

7.5.7 Results

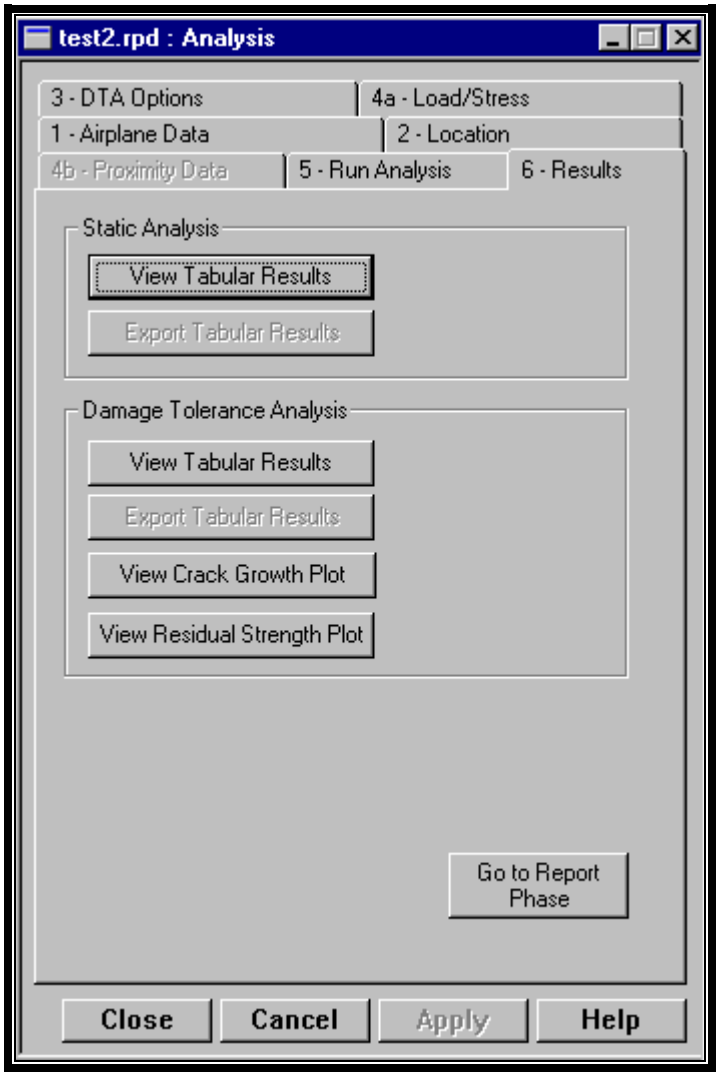


Figure 7-27

From Results page, the user can review the **Static Analysis** and **Damage Tolerance Analysis Output** in both **Tabular** and **Plot** form.

7.5.7.1 Static Analysis Tabular Results

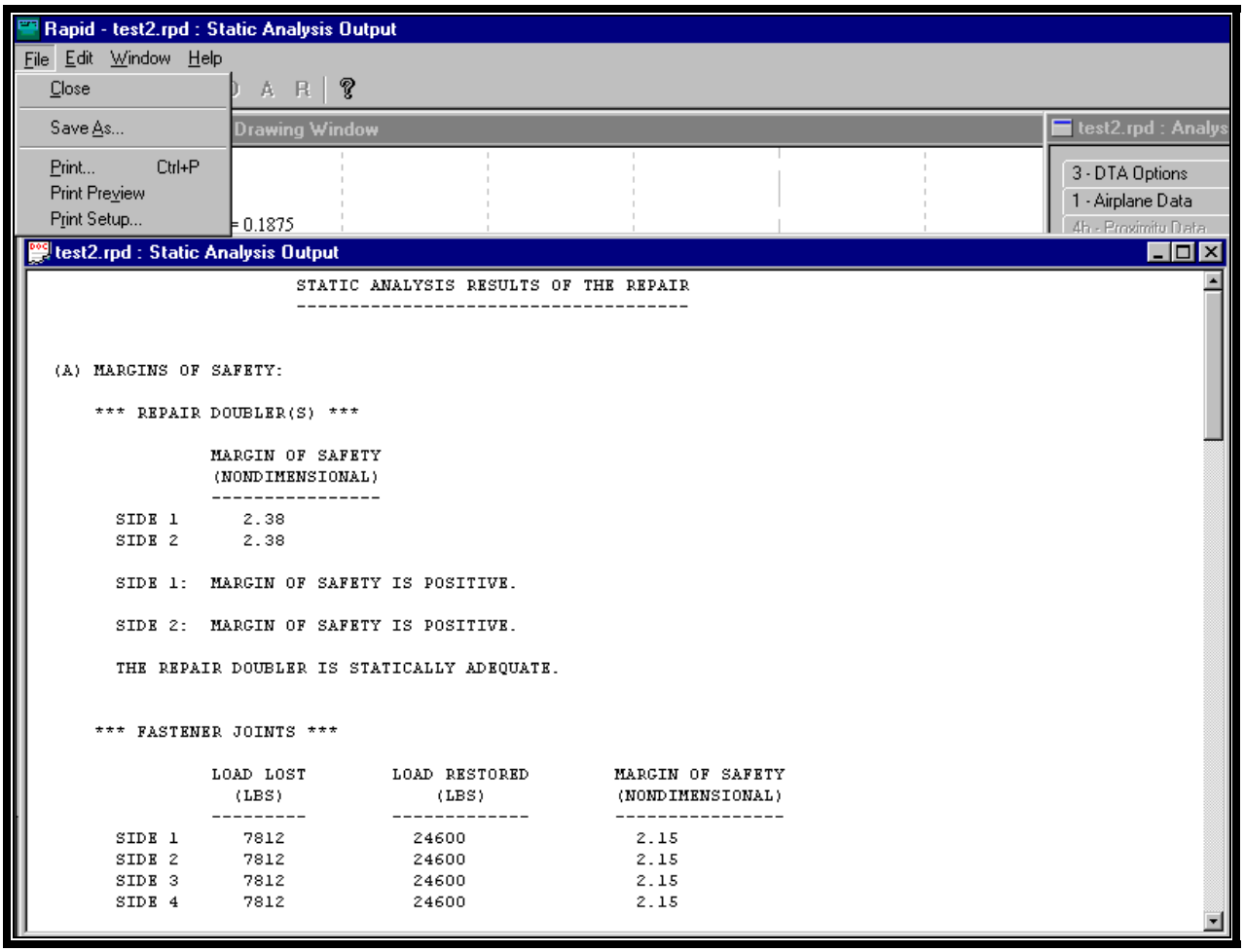


Figure 7-28

This window displays the **Static Analysis Output** from the damage tolerance analysis for all four sides of the repair. This window can be invoked from the **Analysis | Results** page.

To print this data, select the print button on the icon bar, or select the **Print** menu item within the **File** menu. The **Static Analysis Output** window must be the active window to be printed. The user can also **Copy** this window to the Windows clipboard for inclusion into another document --- Pasted.

7.5.7.2 Damage Tolerance Analysis Tabular Results

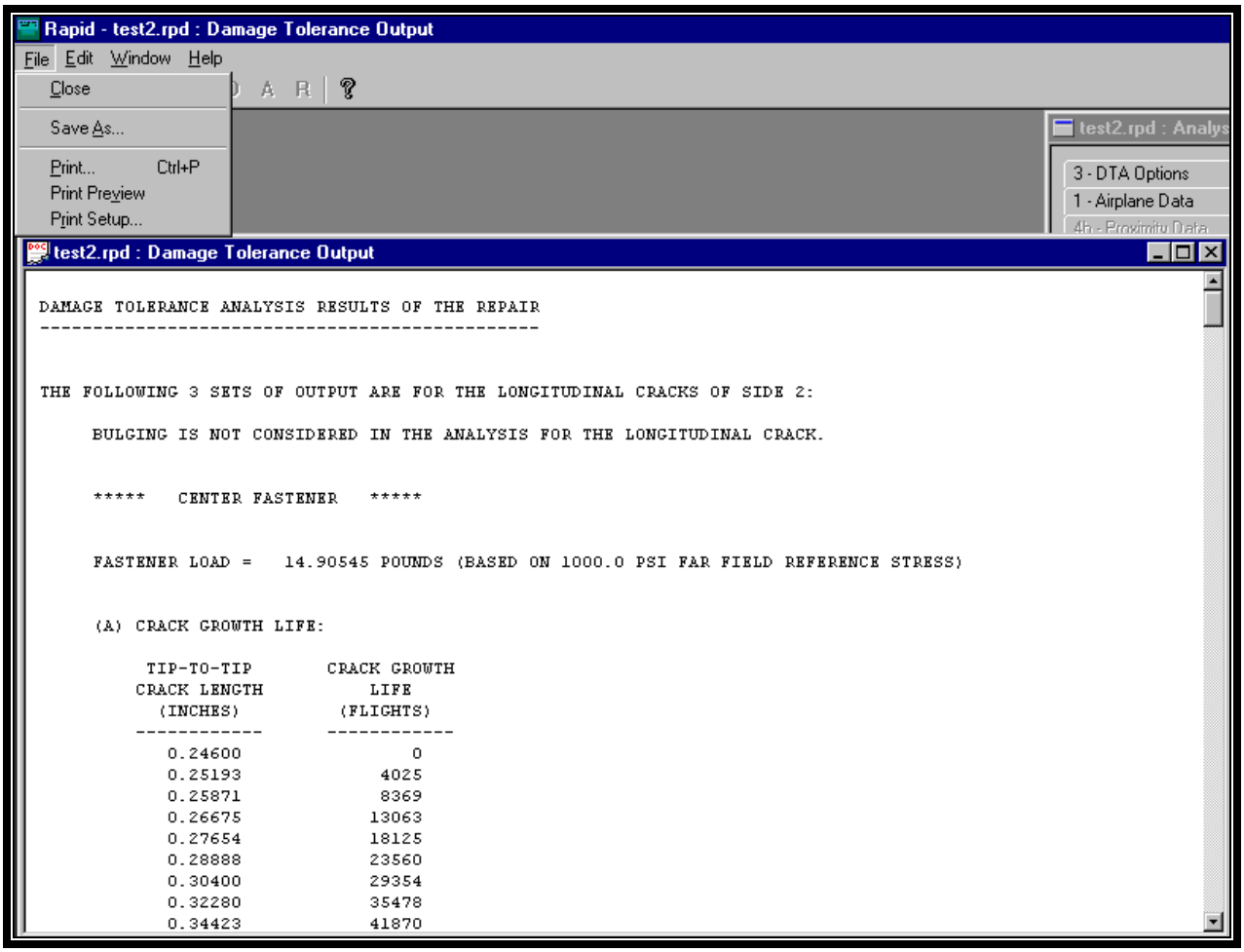


Figure 7-29

This window displays the **Damage Tolerance Output** from the damage tolerance analysis for all four sides of the repair. (For a symmetrical repair, only sides 2 and 3 are available). This window can be invoked from the **Analysis | Results** page.

To print this data, select the print button on the icon bar, or select the **Print** menu item within the **File** menu. The **Damage Tolerance Output** window must be the active window to be printed. The user can also **Copy** this window to the Windows clipboard for inclusion into another document --- Pasted.

7.5.7.3 Crack Growth Plot

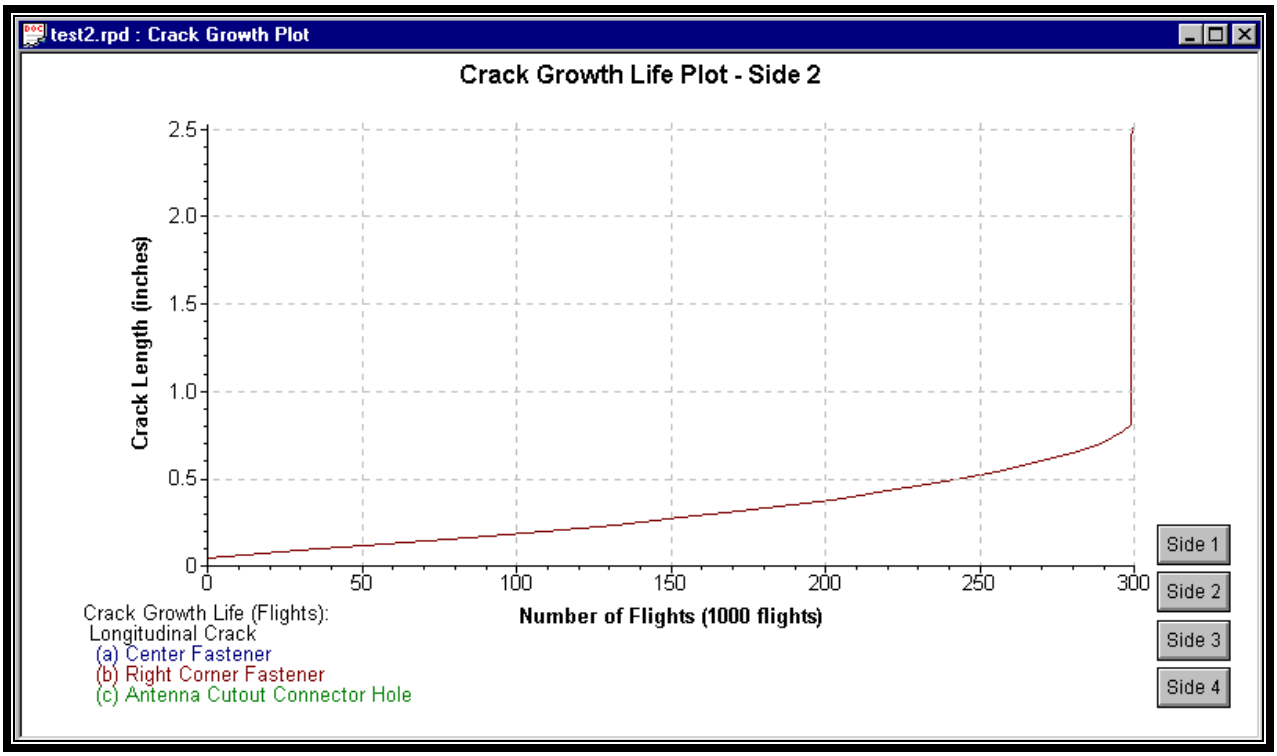
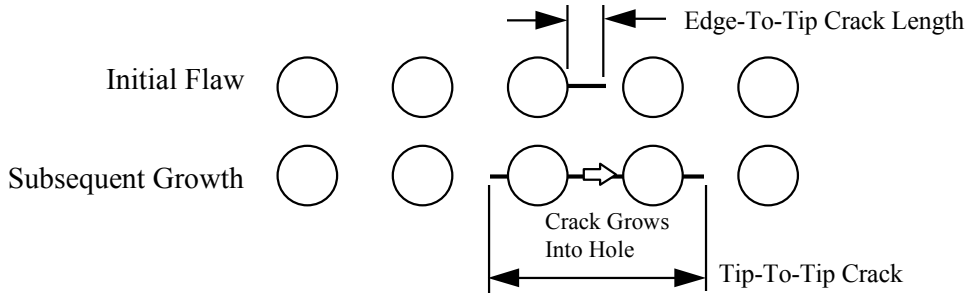


Figure 7-30

This window displays the crack length versus number of flights plot from the damage tolerance analysis for all four sides of the repair. (For a symmetrical repair, only sides 2 and 3 are available.) The predicted crack growth life at each of the critical fasteners is indicated in the legend key. This window can be invoked only from the **Analysis | Results** page. The plots can be generated for longitudinal cracks in the center and corner fasteners above and below the cutout and for circumferential cracks in the center and corner fasteners to the left and right of the cutout. If there are four points, then the plot will not be generated and a message box will be displayed.

The user can choose which side to view. If there is insufficient data to produce a Plot, the user will be notified with an appropriate message.

The crack grows edge to tip and then tip to tip as defined in the figure below.



To print the plot, select the print button on the icon bar, or select the **Print** menu item within the **File** menu. The Plot window must be the active window to be printed. The user can also **Copy** the **Plot** to the Windows clipboard for inclusion into another document --- Pasted.

7.5.7.4 Residual Strength Plot

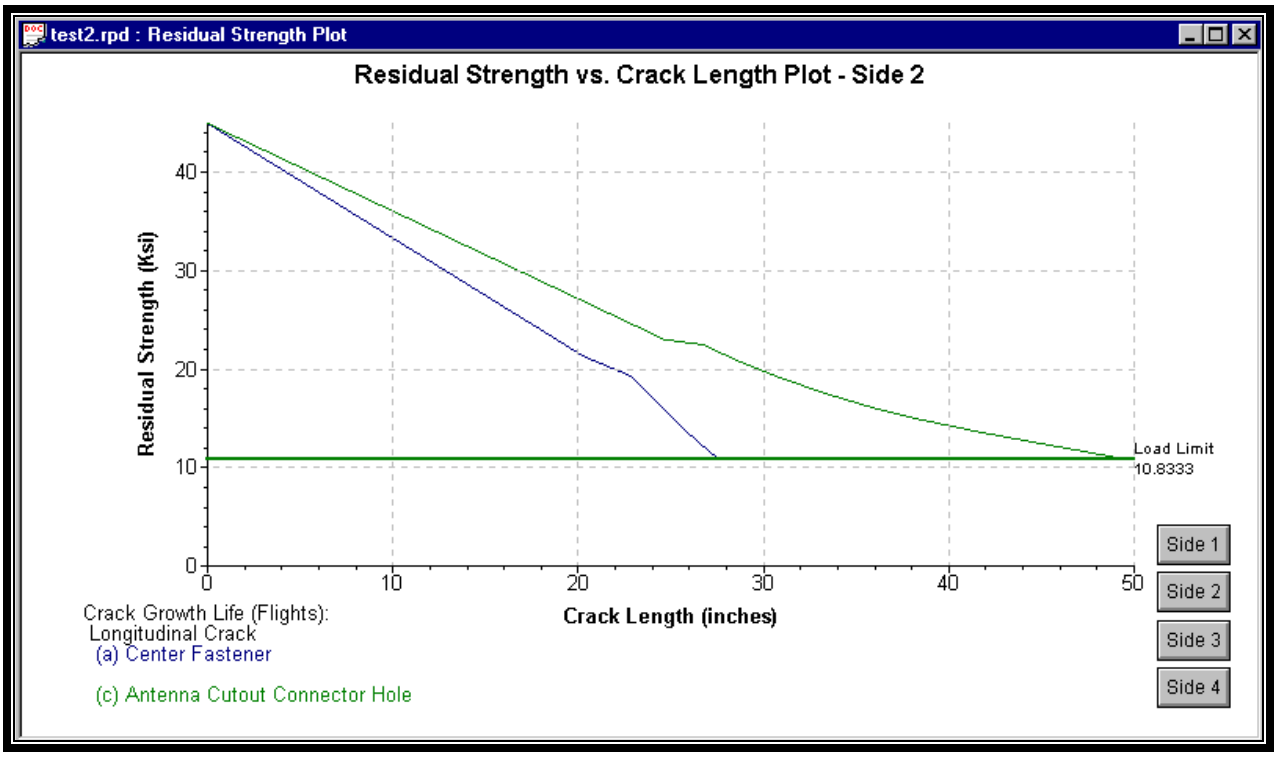


Figure 7-31

This window displays the residual strength versus crack length plot from the damage tolerance analysis for all four sides of the repair. (For a symmetrical repair, only sides 2 and 3 are available). The yield strength of the skin, F_{ty}, is indicated by the solid horizontal line and the **Load Limit** is indicated in the legend key. This window can be invoked from the **Analysis | Results** page. The plots can be generated for longitudinal cracks in the center and corner fasteners above and below the cutout and for circumferential cracks in the center and corner fasteners to the left and right of the cutout. If there are less than four points, then the plot will not be generated and a message box will be displayed.

The user can choose which side to view. If there is insufficient data to produce a Plot, the user will be notified with an appropriate message.

To print the plot, select the print button on the icon bar, or select the **Print** menu item within the **File** menu. The Plot window must be the active window to be printed. The user can also **Copy** the **Plot** to the Windows clipboard for inclusion into another document --- Pasted.

7.6 Report Property Sheet

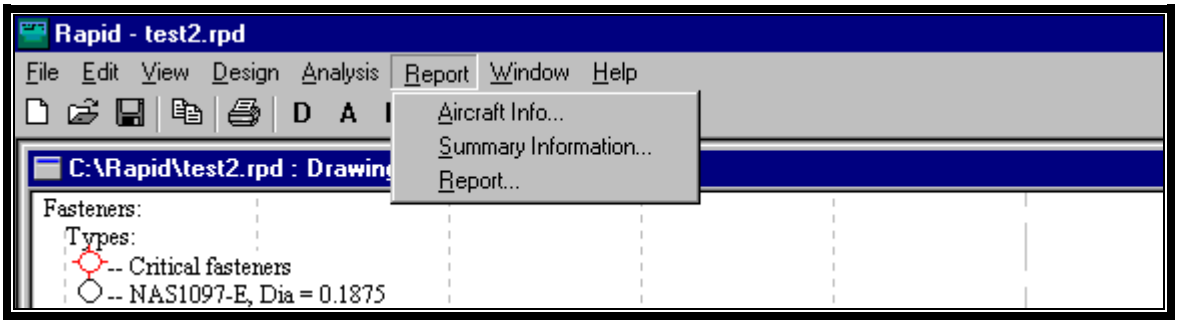


Figure 7-32

This activates the **Report** Property Sheet containing the pages to create a design. On a new design, all pages are initially available, as these pages only affect the Report.

7.6.1 Aircraft Info

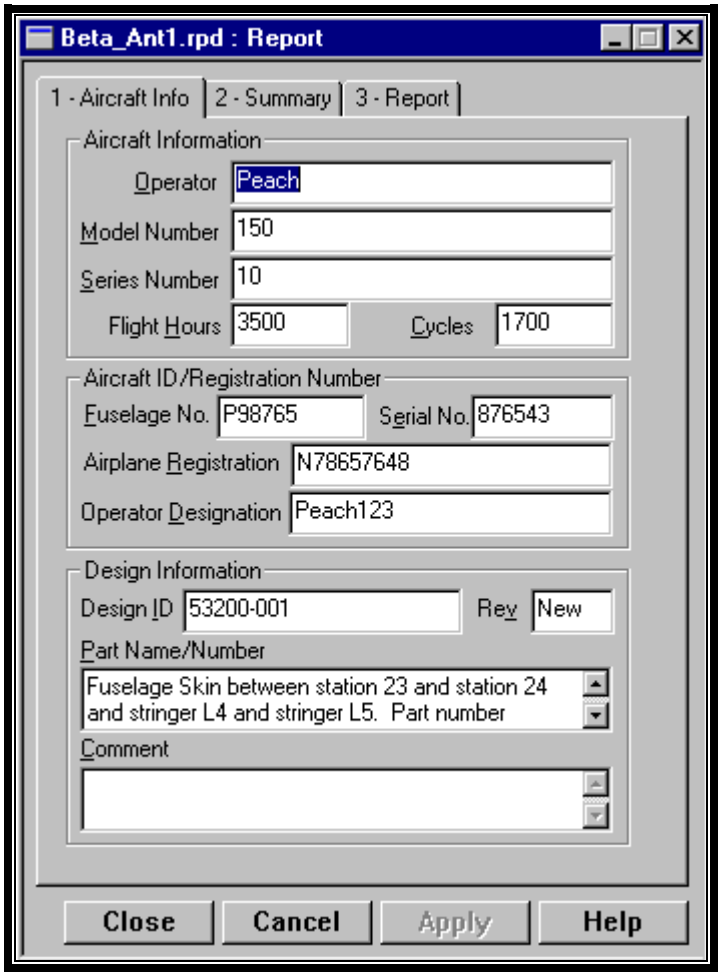


Figure 7-33

This page collects the information used for the first page of the **Report**.

If no information is supplied for a particular field, the entry "No information supplied" will appear on the Report for that field.

7.6.2 Summary

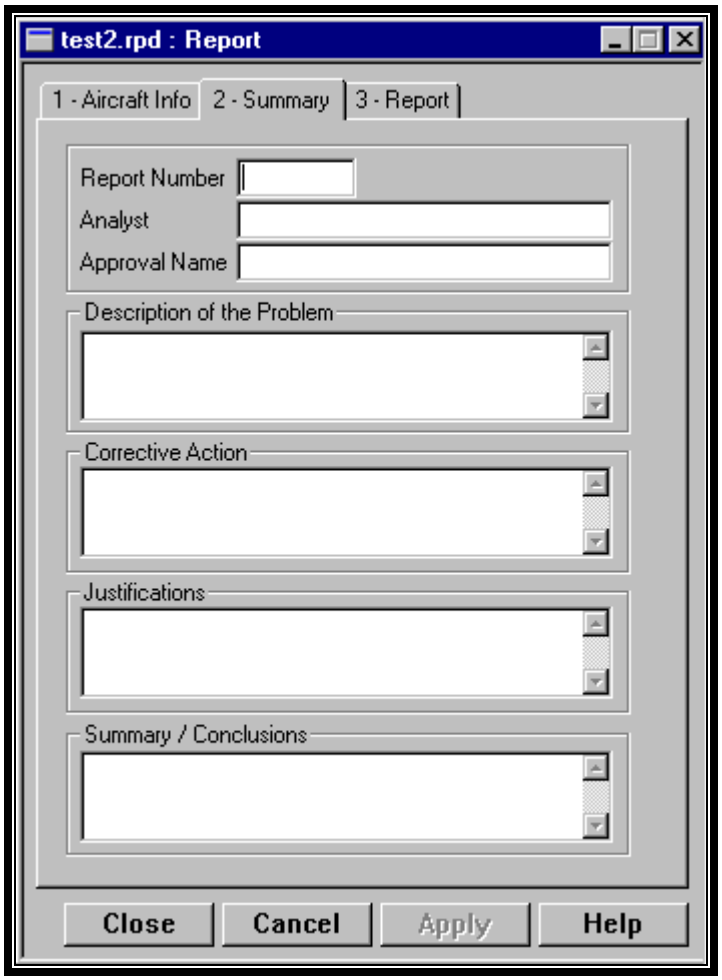


Figure 7-34

This page allows the user to supply more information for the Report.

If no information is supplied for a particular field, the entry "No information supplied" will appear on the Report for that field.

The bottom 4 fields are multi-line edit fields to be used as mini editors. The maximum number of characters for each of these fields is limited to 512 – including spaces, punctuation, and carriage returns.

7.6.3 Report

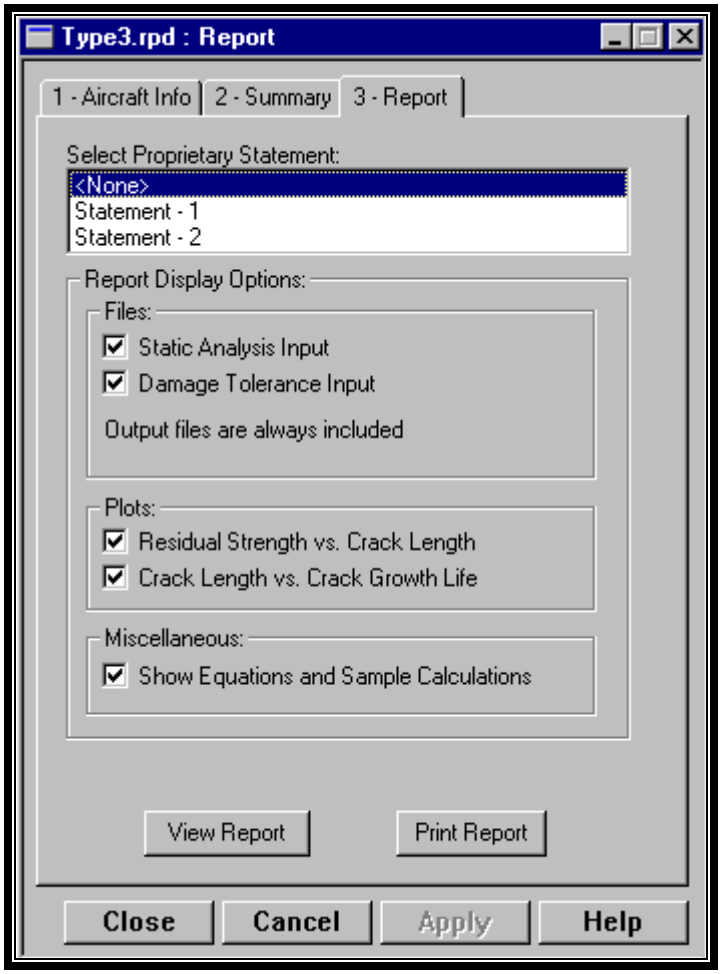


Figure 7-35

This page allows the user to choose which additional items will appear on the **Report**.

Select Proprietary Statement allows the user include a statement on the **Report**. Picking <None> bypasses the printing of the statement. Statements can be added in using any of 2 methods --- (1) Double-click the right mouse button in the **Statement Box** will bring up the **Proprietary Statement Dialog**. (2) The **Proprietary Statement Dialog** can be accessed directly by choosing the **File | Program Setup** submenu.

The **View Report** allows the user the check the **Report** prior to printing. It also provides the user with the ability to save the **Report** for inclusion in another document. Printing the **Report** from this view will NOT include the Drawing and Plot views.

The **Print Report** sends the **Report** to the printer and "attaches" the Drawing and Plot views to the end of the Report.

Changes in the Report options disables the **View Report** and **Print Report** buttons until the user **Applies** those changes.

Close the property sheet by mouse clicking the "Close" button

8. References

1. Military Handbook, "Metallic Materials and Elements for Aerospace Vehicle Structures," Wright-Patterson AFB 45433-6533, MIL-HDBK-5F, November 1990.

2. Swift, T., "Fracture Analysis of Stiffened Structure," Damage Tolerance of Metallic Structure: Analysis Methods and Application, ASTM STP 842, J.B. Chang and J.L. Rudd, Eds., American Society for Testing and Materials, 1984.

3. Swift, T., "Repairs to Damage Tolerant Aircraft," Structural Integrity of Aging Airplanes, S.N. Atluri, S.G. Sampath, and P. Tong, Eds., Springer-Verlag, 1991.

9. Appendices

9.1 Appendix A -- Material Database

The **Database** menu section of the **Tools** menu contains **Materials** and **Fasteners** menu selections, which are described in the following subsections.

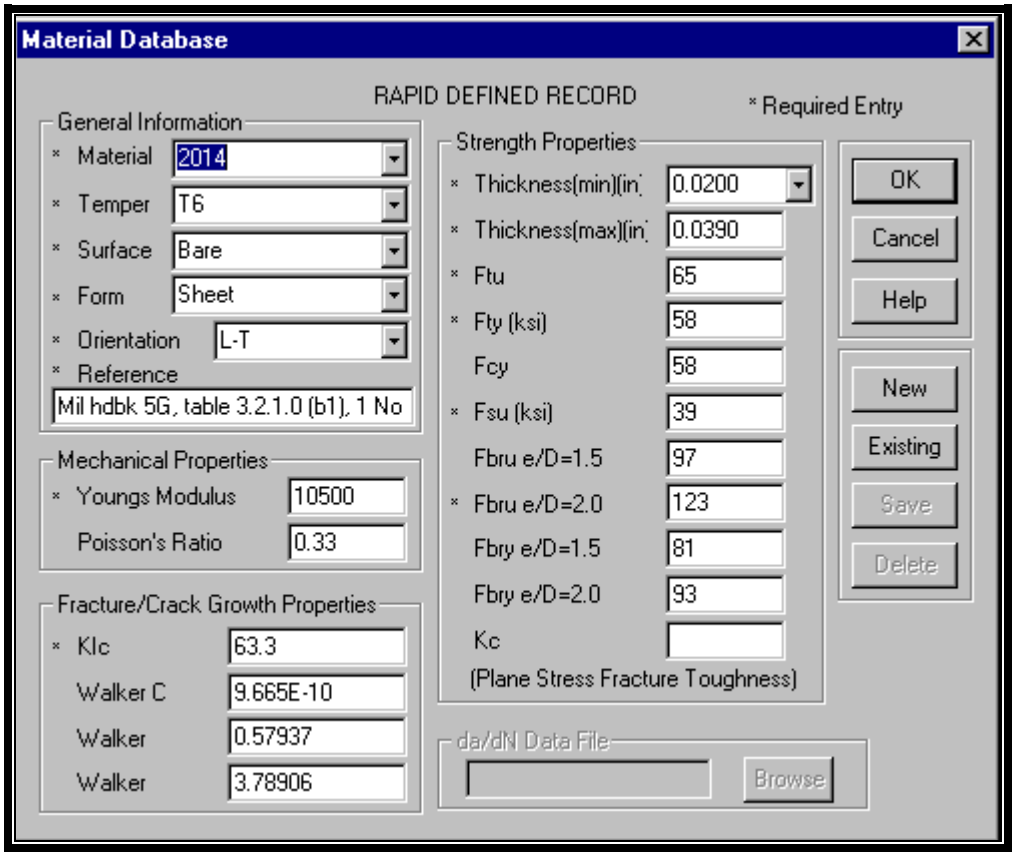


Figure 9-1

This window is used to review and add to the database of materials available for analysis. There is no icon button on the button bar of the main RAPID MDI window for this dialog window; it must be opened from the **Database** menu item in the **File** menu.

The text in the upper right of the dialog indicates whether the current material is a RAPID-defined record or a user-defined record. To add a new material, depress the **New** button. This will clear the entries to allow user-defined data to be entered, which can then be saved by selecting the **Save** button or discarded by selecting the **Existing** button to return to RAPID defined material or the **Cancel** button. To edit a user defined material, type the change and depress the **Save** button. The **OK** button closes the window.

Only user-defined records can be deleted with the **Delete** button. The **Delete** dialog also allows the user to delete entries for a particular Thickness.

The terms are defined as follows:

K_{Ic} = Plane strain fracture toughness

C, q, p = Coefficients of the Walker crack growth rate equation

$$\frac{da}{dN} = C\left[(1-R)^q K_{max}\right]^p$$

where

K_{max} = the stress-intensity factor associated with the maximum stress in the stress spectrum.

F_{tu} = Allowable tensile stress

F_{ty} = Allowable tensile yield stress at permanent strain = 0.002

F_{cy} = Allowable compressive yield stress at permanent strain = 0.002

F_{su} = Allowable ultimate stress in pure shear (this value represents the average shearing stress over the cross-section)

F_{bru} = Allowable ultimate bearing stress

e/D = Ratio of edge distance to hole diameter

F_{bry} = Allowable bearing yield stress

K_c = Critical plane stress fracture toughness, a measure of fracture toughness at point of crack growth instability

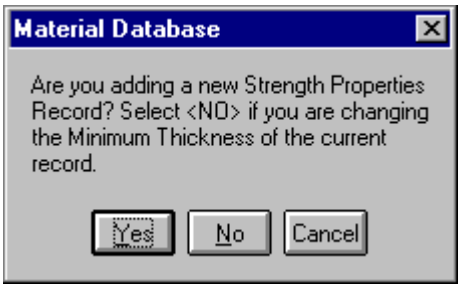
If K_c is not present, then it will be automatically calculated using the NASA/FLAGRO 2.0 equation:

$$\frac{K_c}{K_{Ic}} = 1 + B_k e^{-(A_k t/t_0)^2}$$

where

$t_0 = 2.5\ (K_{Ic}/F_{ty})^2$

When adding a new entry, if the **Thickness (min)** field is overwritten and then that field is exited by pressing the tab key or selecting another field with the mouse, then the following dialog window will appear. This allows the user to choose between adding properties at a new minimum thickness or changing the value of the minimum thickness for the current record



When the **Delete** button is selected, the following dialog appears to allow the user to choose between deleting the entire record or simply the material strength properties for the displayed material thickness.

9.2 Appendix B -- Fastener Database

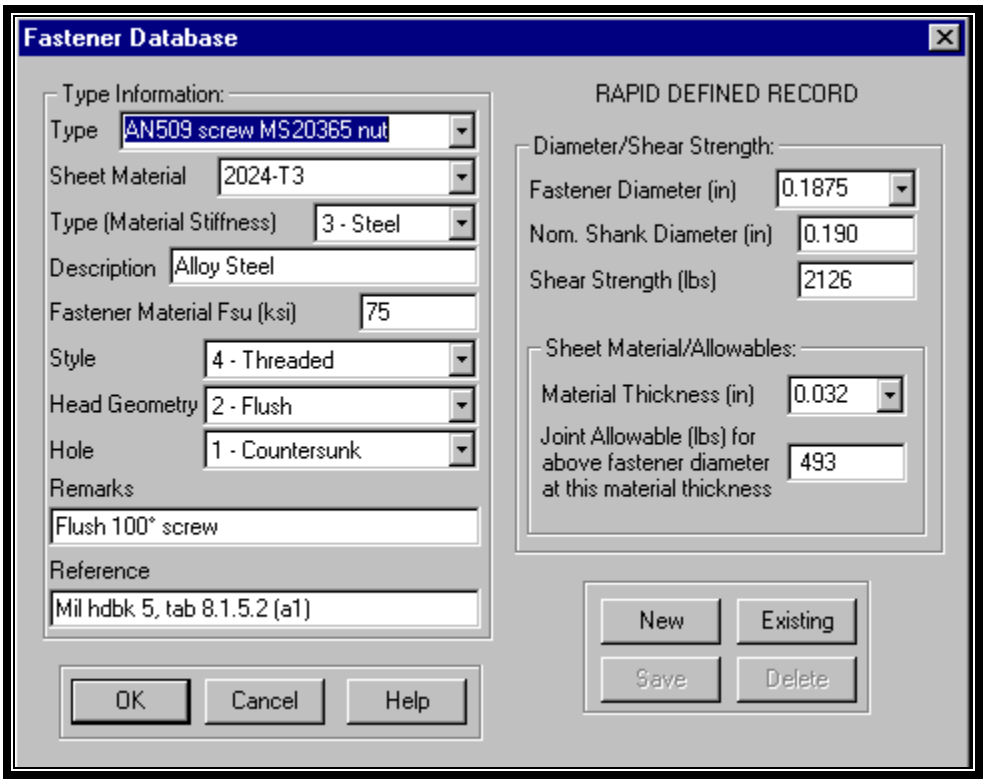
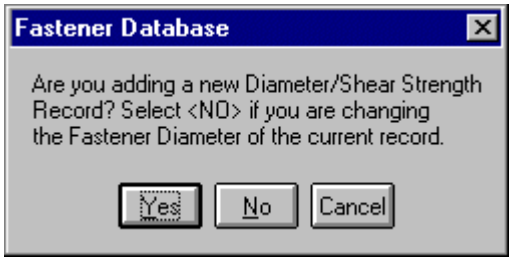


Figure 9-2

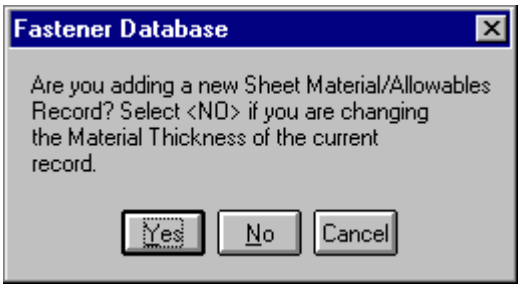
This window is used to review and add to the database of fasteners available for analysis. There is no icon button on the button bar of the main RAPID MDI window for this dialog window; it must be opened from the pull-down **Database** menu item in the **File** menu.

To add a new material, depress the **New** button. This will clear the entries to allow user-defined data to be entered, which can then be saved by selecting the **Save** button or discarded by selecting the **Existing** button to return to RAPID defined fastener or the **Cancel** button. To edit a user defined fastener, type the change and depress the **Save** button. The **OK** button closes the window.

When adding a new entry, if the **Fastener Diameter** field is overwritten and then that field is exited by pressing the tab key or selecting another field with the mouse, then the following dialog window appears. This allows the user to choose between adding properties at a new fastener diameter or changing the value of the fastener diameter for the current record.



Similarly, when adding a new entry, if the **Material Thickness** field is overwritten and then that field is exited by pressing the tab key or selecting another field with the mouse, then the following dialog window appears. This allows the user to choose between adding properties at a new material thickness or changing the value of the material thickness for the current record.



When the **Delete** button is selected, the following dialog allows the user to choose between deleting the entire record, the portion of the data associated with the displayed diameter, or the portion of the data associated with the displayed material thickness.

9.3 Appendix C -- DTA Options – Advanced User

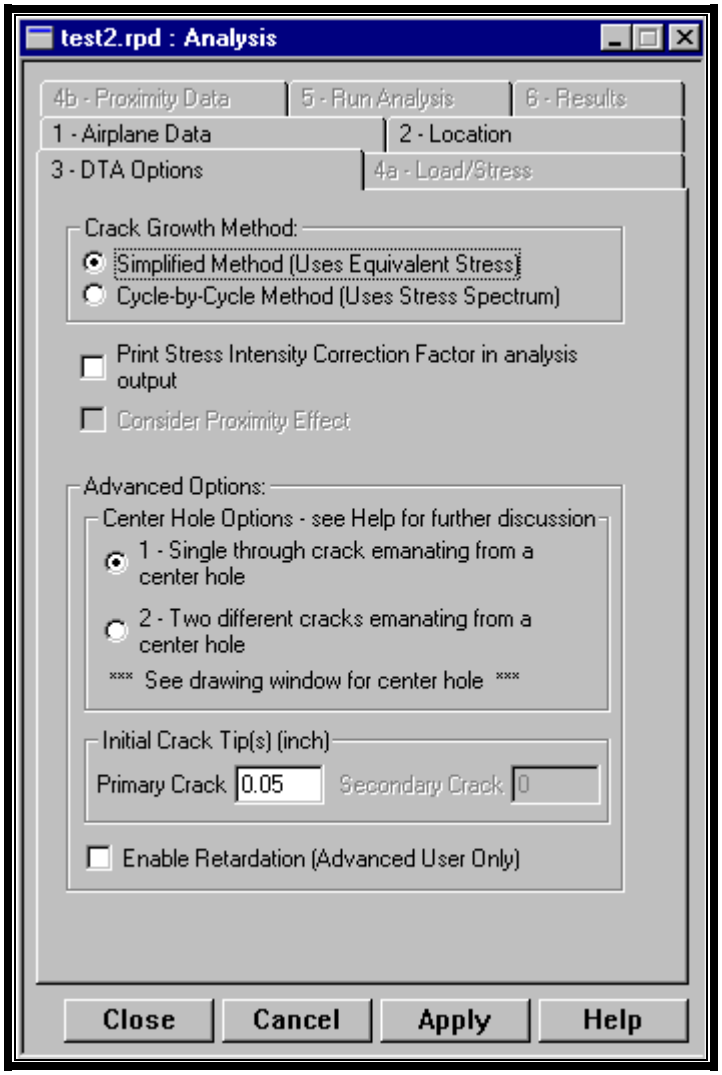
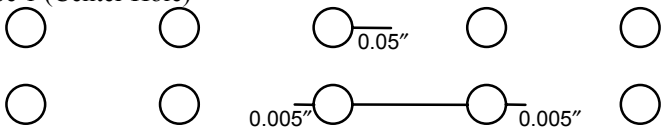


Figure 9-3

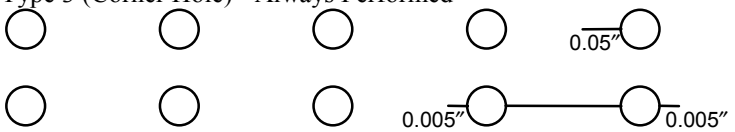
The user may select between two **Center Hole Options** for the crack growth along with a corner hole scenario, as defined below:

a. Scenario Type 1 (Center Hole) -



b. Scenario Type 2 (Center Hole) -

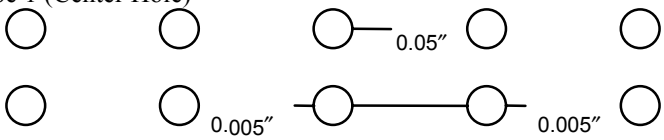
c. Scenario Type 3 (Corner Hole) - Always Performed



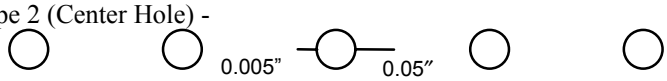
For the current version, the program is restricted to assuming a non-visual inspection method. The crack growth method and retardation flags are set as documented in the Analysis Methods Document.

The crack growth scenarios are defined as follows:

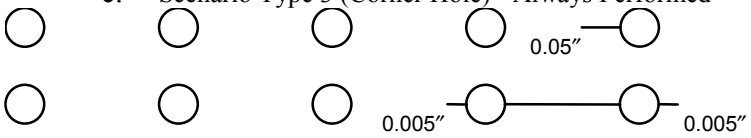
a. Scenario Type 1 (Center Hole) -



b. Scenario Type 2 (Center Hole) -



c. Scenario Type 3 (Corner Hole) - Always Performed



9.4 Appendix D -- Description of Analysis Input Files

The RAPID analysis programs are compiled FORTRAN DLL Applications:

Static Analysis	- REPAIRS.DLL
Damage Tolerance	- REPAIRD.DLL

Both DLLs read "initialization files" – these files contain the input, output, error, and database filenames that are read by the analysis programs (the following 2 filenames are hard-coded into the program) :

Static Analysis	- REPAIRS.@#$
Damage Tolerance	- REPAIRD.@#$

Obtain the Data

The Static Analysis input is stored in the [~STA_INP] section of the saved repair configuration.

1. Open the saved file using any ASCII text word processor
2. Search for [~STA_INP]
3. Block this section and copy it to the clipboard
4. Paste from the clipboard to your input file

Example of the start of the section:

```
    :
;[~STA_INP]
REPAIR CONFIGURATION TYPE (1 External Doubler)
             3
SKIN/DAMAGE : Number of Layers
    :
```

The Damage Tolerance Analysis input is stored in the [~DTA_INP] section of the saved repair configuration.

Use the above procedure to cut and paste to your document.

Example of the start of the section:

```
    :
;[~DTA_INP]
4

REPAIR CONFIGURATION TYPE (1 External Doubler)
             3
      SKIN: Mat. Type
```

NOTE -- The input decks for both the static and damage tolerance files must NOT contain any blank lines.

The binary stress spectrum files used by damage tolerance analysis can be found by searching for the [~RPD_SP0] and [~RPD_SP1] sections of the saved repair configuration. The entries in these sections give the files names of the stored data.

Example of the sections:

```
;[~RPD_SP0]
TYPE31._S0

;[~RPD_SP1]
TYPE31._S1
```

These file names are required for the initialization file described below.

Modify the Initialization File

STATIC Analysis

The static analysis DLL, REPAIRS.DLL, reads the initialization file, REPAIRS.@#$
File specification for REPAIRS.@#$

```
repairs.inp            // input deck created by RAPID GUI
repairs.out            // output deck created by REPAIRD.EXE
repairs.err            // error file created by REPAIRD.EXE
```

These files can be named anything you wish, the program names them:

```
repairs.inp            ~sta_inp.<#>
repairs.out            ~sta_out.<#>
repairs.err            ~sta_err.<#>

The extension, <#>, is an integer the program assigns when it opens a file or starts a
new file.
```

DAMAGE TOLERANCE Analysis

The damage tolerance analysis DLL, REPAIRD.DLL, reads the initialization file, REPAIRD.@#$

File specification for REPAIRD.@#$

```
repaird.inp              // input deck created by RAPID GUI
repaird.out              // output deck created by REPAIRD.EXE
repaird.err              // output deck created by REPAIRD.EXE
repaird.sif              // *beta factors for baseline repairs
repaird.mat              // *material database factors
repaird.fst              // *skin stress factors for baseline repairs
1         // flag to indicate the next 2 files are the spectrum files to be used
repaird.sp0              // binary spectrum file(circumferential stress)created by RAPID GUI
repaird.sp1              // binary spectrum file(longitudinal stress) created by RAPID GUI
1         // flag to indicate the next file is the proximity file to be used
repaird.prx              // *fastener load transfer factors due to proximity effect
1         // flag to indicate the next file is the splice joint file to be used
repaird.jnt              // *skin stress factors for splice joint repairs
1         // flag to indicate the next 3 files are the stiffener files to be used
repaird.stf              // *skin stress factors due to stiffener effect
repaird.stb              // *stiffener beta factors
repaird.lcf              // *life correction factor database for splice joints
1         // flag to indicate the next file is the circular file to be used
repaird.cir              // *circular repair ratio factors
```

Note: 0=Do not read filename on next line; 1=read filename on next line.
* Do not change these file names

User may execute analysis with either proximity effect, stiffener effect, or splice joint. These effects cannot be combined yet. So only **one** of the flags may be set to 1.

The following files can be named anything you wish, the program names them:

```
repaird.inp              ~dta_inp.<#>
repaird.out              ~dta_out.<#>
repaird.err              ~dta_err.<#>
repaird.sp0              ~rpd_sp0.<#> // binary spectrum file - circumferential stress
repaird.sp1              ~rpd_sp1.<#> // binary spectrum file - longitudinal stress

The extension, <#>, is an integer the program assigns when it opens a file or starts a new
file.
```

9.5 Appendix E -- Description of RAPID.INI File

The RAPID.INI file is used to provide initialization and default data for the application.

It is structured like a generic Windows' INI file. The text within the brackets [..] is called the "section" and the text under each "section" are called "entries." RAPID accesses and updates this file through the Tools | Setup menu. RAPID is shipped without this INI file. With the initial run of RAPID, an INI file is constructed with hard-coded data and file location entries on the current drive/directory.

The section/entry names **must not** be changed!!!

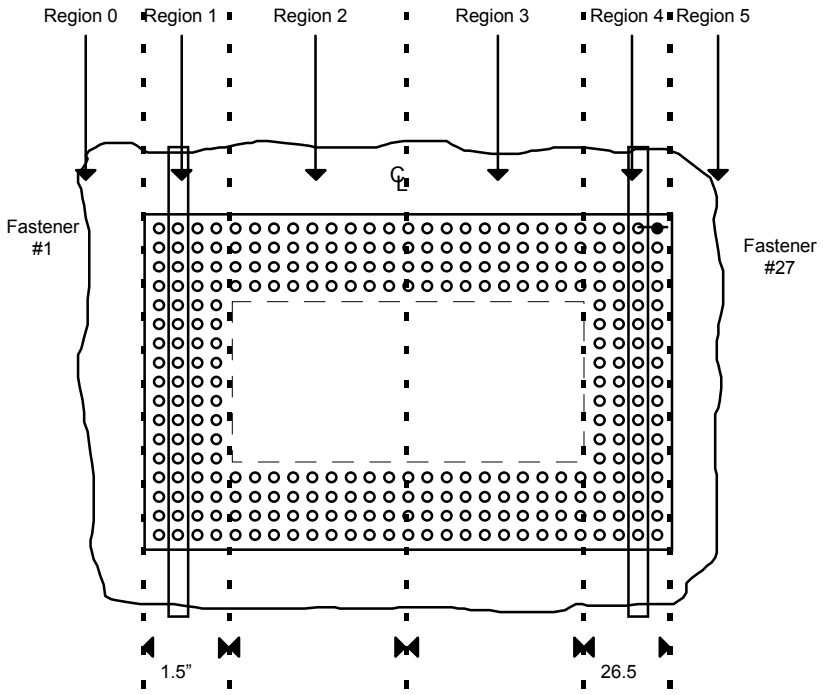
```
[Version]
Number=1.2

[File_Locations]
Prog_Dir=D:\RAPID2C
Data_Dir=D:\RAPID2C
Temp_Dir=D:\RAPID2C
Loads_Dir=D:\RAPID2C\LOADS

[RepairType]
Type=TYPE30
ProximityFlag=0   // 0-false    1-true
StiffenerFlag=0   // 0-false    1-true
LapReversedFlag=0 // 0-false    1-true

[Report_Options]  // parts to be printed on the report
Static_Input=0    // 0-false    1-true
Static_Results=1  // 0-false    1-true
DTA_Input=1       // 0-false    1-true
DTA_Results=0     // 0-false    1-true
Residual_Plot=0   // 0-false    1-true
Flights_Plot=0    // 0-false    1-true
Inspection_Plot=0 // 0-false    1-true
Analysis_Notes=0  // 0-false    1-true

[Fastener_Data]
Long_Pitch=1.0
Circum_Pitch=1.0
Long_Edge=0.5
Circum_Edge=0.5
Style=NAS1097-E
Diameter=0.1875
Rows=3

[DTA_Options]
Run_Type=2
Crack_Growth=1
Hole_Options=1
Visual_Method=0   // 0-false    1-true
Retardation=0     // 0-false    1-true
PrimaryTip=0.05
SecondaryTip=0.0
OperatingPressure=
```

```
[Stress_Data]
EquivStress=1     // 1-RAPID calculated with RAPID data
                  // 2- User entered value
                  // 3- RAPID calculated with User data
EquivStressValue= // Equivalent Stress value
SpectrumUsed=3    // 1-RAPID defined 2-User defined (binary)
                  // 3-User defined (ascii)
SpectrumName=Generic Narrow Body
InputFormat=1     // 1-Pseudo-Flights (repeatable flight sequence)
                  // 2-Cycle-by-Cycle Stresses (repeatable block)
RainFlow=0        // 0-false    1-true
Truncation=0      // 0-false    1-true
TruncRange=2.0
```

9.6 Appendix F -- Description of Stiffener Effects Regions

Damage Tolerance Input File

```
              .
              .
              .
              .
              .
STIFF    : MAX #, POS,   CSA,   YNG.MOD,      HPIT
           0      0.0    0.0    0.0           0.0
           1      1.5    0.5    10500         0.909
           0      0.0    0.0    0.0           0.0
           0      0.0    0.0    0.0           0.0
           1      26.5   0.5    10500         0.909
           0      0.0    0.0    0.0           0.0
              .
              .
              .
```



Regions -Each repair has 6 regions numbered from left to right:

Region 0: Outside left (repair boundary) of the repair to the left most side of the repair (absolute distance)
Region I: Left most side of the repair to the left most side of the cutout
Region II: Left most side of the cutout to the center line of the repair
Region III: Center line of the repair to the right most side of the cutout
Region IV: Right most side of the cutout to the right most side of the repair
Region V: Outside right (repair boundary) of the repair to the right most side of the repair

Stiffener Input Line fields:

MAX # Number of stiffeners in region; maximum of 8 stiffeners per region

POS Absolute position of the stiffeners in relation to the whole repair (from the left edge of the repair); for:

REGIONS 1-4, $0.0 \leq POS \leq Max.DoublerSize$;
REGION 0, POS is the absolute distance of the stiffener away from the left edge of the repair;
REGION 5, POS is the absolute distance of the stiffener away from the right edge of the repair.

CSA Cross-sectional area of the stiffener

YNG.MOD Young's modulus of the stiffener

HPIT Pitch of the fasteners in the stiffener

DBLR ID - If more than 1 repair doubler, indicate which repair doubler is being used in referencing the stiffener's positions, 1=first doubler and 2= second doubler. If there is only 1 repair doubler, then DBLR ID=1.

Note: Each stiffener in a region has a position, cross-sectional area, Young's modulus, and fastener pitch in the stiffener parameters associated with it.

Although there might not be any stiffeners in the repair or region, enter *0* (or *0.0*) in the number of stiffeners, position, cross-sectional area, Young's modulus and pitch parameters.